Aerodynamic Design of Formula Student Car

.

Author: *Daniel Gómez Lendínez*
Tutor: *Dr. Daniel García-Pozuelo Ramos*

Bachelor in *Mechanical Engineering*

Department of *Mechanical Engineering*
University Carlos III of Madrid

July 2013

Leganés Madrid

Abstract

This bachelor thesis analyzes a solution for the aerodynamic problem in a Formula Student car. This solution will be implemented in a real car and in a real competition with the purpose of get a good performance in the race.

The project study the critical parts of a formula car in terms of drag and lift, which are the nose, the air flow for cooling and the diffuser. This design will be taking into account due to the characteristics of the circuit and the competition the maximum speed will be between 80 and 100 kilometers per hour.

The construction and simulation will be with SolidWorks software and assumming a sunny day, and sea level pressure. Also it will be assumed a speed of 25 meter per seconds that are 90 kilometers per hour.

SolidWorks software can simulate Computer Fluid Dynamics that show the performance of the whole car in terms of drag and lift and Finite Element Method to analyze the structure in terms of strength of materials.

Acknowledgements

I want to say thank you:

To *Dr. Daniel García-Pozuelo Ramos* and *Dr. Antonio Gauchía Babé* for their time and patient in the constitution and daily work in the Formula UC3M Team.

To *Dr. Eduardo Antonio Fernández Tarrazo* because with his initial help to understand fluid dynamics this project will not exist.

To *Dr. Javier Rodríguez Laguna* and *Dr. Silvia Noemí Santalla Arribas* for encouraging the students to start with the team and show freshmen students that university is more than study and passing courses.

To *Javier López Carrillo* and *Jorge Vega Hernández* because without all the time and work that they dedicate to the team, this thesis will not be possible.

To *Olalla Bravo Conde* for listening and helping me.

To *Daniel Moreno Cenjor* for his ideas from other point of view.

To my *family* and some *friends* that supported me during the degree and is impossible to mention everyone.

Contents

List of Tables

List of Figures

Chapter 1

Introduction

1.1 Formula Student

The Formula Student also known as European Formula SAE [11] is a competition between undergraduates students who develop a formula car that they design, construct and drive as small Formula 1 team. The students get a real experience in design and construction, they learn how to work in teams and under pressure. The competition is organized by the SAE (Society of Automotive Engineers) [12] in USA and by Institution of Mechanical Engineers (IMechE) in UK [13] . The figure 1.1 shows the logo of the first European competition.

Figure 1.1: Formula Student logo

The first competition of Formula SAE was in 1981 when the Texas University organized an event with six teams and forty students in Austin. After that, in 1998, it comes to Europe, and the first time was in the United Kingdom. Now there are competitions in Germany whose circuit can be seen in figure 1.2 , Brazil, Japan and also Spain, this one, organized by *Sociedad de Técnicos de Automoción (STA)* [14]. In each event they can be up to 120 teams and 2000 students.

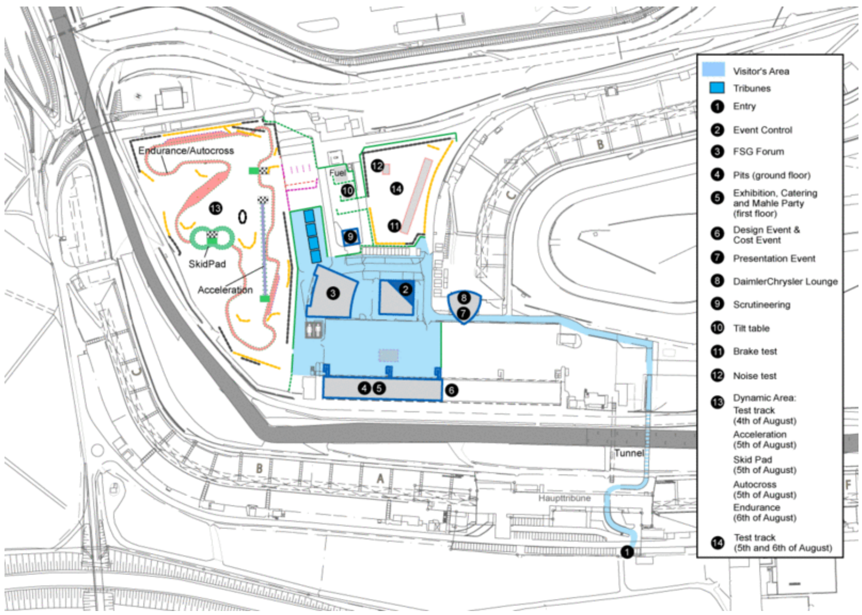


Figure 1.2: Formula Student Germany circuit

The goal of this competition is to simulate a real situation in which a hypothetical company design a prototype for a race. The supposed buyers will be amateur drivers that need a car and a team structure to win a race. The car must meet high requirements in acceleration, brake and stability but also has to be cheap, easy to maintenance and reliable. Another factors are also taking into account like aesthetic and comfort. The maximum price for the vehicle will be 21,000 dollars that are approximately 16,500 euros.

In the competitions of Formula Student there are different categories:

- Fuel engines class: typical cars with combustion engines limited to 610 cubic centimeters but not in type of fuel that can be for instance gasoline or GLP.
  - Class 3: new teams that have cars in the design and validation step. A team can just participate in this category one time. They can only get points from design, business and cost events because they have not any car builded.
  - Class 2: this category is for teams that they made a chassis but they cannot run with it. Also they can only get points from the static events and maximum time is limited to one year.
  - Class 1: the teams have a car that can run and its participation is limited to two times if they are consecutive. This rule, force the teams to made a new chassis and innovate in the design. The teams get points in all events.
- Electric or hybrid class: the participants have their own rules because of the weight and electric safety conditions.

1.1.1 Competition

The Formula Student event consists in a series of static and dynamic testing operations that the teams have to complete the engineering project of creating a race car, with its performance, its safety behavior and the ability of the workteam to pass all the activities. [Annex Rules]

- Static Test
  - Technical verification: be sure that the car fulfill all the safety conditions and regulations.
  - Design: the team has to defend the project in front of four judges in a first step, and only fourteen teams go to the final which consist in four hours in front of thirty judges.
  - Business and presentation: it consist in a explanation of how you are selling your car in fifteen minutes in front of marketing professionals.

  - Cost analysis and sustainability: the team discuss with two judges a report of the cost of the car, including pieces and labor.

- Dynamic Test

  - Acceleration: the car has to go over 75 meters in the less time as possible, being the maximum time 5.8 seconds. Each team has two opportunities to do the event with different drivers. If the driver pulls two cones, the team will be out of this test.
  - Skidpad: the car has to go over two circumferences of 18.25 meters of radii and 3 meters of width forming an eight. It has to do two times in each direction. Pull a cone is equivalent to 0.25 seconds of penalization.
  - Autocross: the circuit for this event has a straight of 60 meters and another one of 45 meters, a curve of radius of 23 meters, and a second one of 45, slalom with cones separated between 7.62 and 12.19 meters, chicanes and variable radii curves. The total length is approximately 800 meters, and the driver has to do a certain number of laps. The penalization is two seconds per cone pulled.
  - Endurance : this event is 22 laps in a circuit with two straight of 77 and 61 meters, slaloms, and different types of curves. The most important in this event is the fiability and the fuel consumption. Two seconds per cone and twenty per slalom done incorrectly is the penalty.

The point distribution change every year, in case of 2013 it can be seen in table 1.1.

Static events	**325**
Design	150
Cost and sustainability	100
Business and presentation	75
Dynamic events	**675**
Skidpad	50
Acceleration	75
Autocross	150
Endurance	300
Efficiency	100
Total	**1000**

Table 1.1: Points in the competition

1.1.2 Escuderia UC3M

Formula UC3M Team are the students from of University Carlos III of Madrid that is currently integrated by members of Escuderia UC3M association [15]. Founded in 2011, they want to construct the first vehicle Formula Student car developed by students of this university with the goal of going to competition in 2014, and motivate students who have interest in convert their theoretical knowledge into a real project.

Figure 1.3: Escuderia UC3M logo

The team currently consists of the following divisions to organize the design process:

- Aerodynamics and bodywork design.
- Chassis.
- Electronics.
- Brakes and steering systems.
- Engine and transmission.
- Suspension system.

All this divisions are coordinated by the technical director, who ensures that the designs can be assembled to the vehicle and meet the requirements. Also, the team has a marketing division that search for sponsors.

1.1.3 Limitations

The aerodynamic elements must fulfill the requirements because during the competition the judges check them. [Annex Rules]

- In plain view, any aerodynamic part can be:
  - No further forward than 762 mm of the fronts of the front tires.
  - No further rearward than 305 mm of the rear of the rear tires.
  - No wider than the outside of the front tires or rear tires measured at the height of the hubs.
- All devices, that can contact a pedestrian must have a minimum radii of 1.5 mm.
- No power device may be used to move or remove air from under the vehicle except fans designed exclusively for cooling. Power ground effects are forbidden.
- The egress from the vehicle have to be less than 5 seconds, and must not require any movement of any element or device.
- The wing or wings must be mounted in such positions, and sturdily enough, that any accident is unlikely to deform the wings or their mountings in such a way to block the drivers egress.

1.2 Goals

A good aerodynamic design has some characteristics that are detailed below:

- Smooth profiles with favorable pressure gradients.
- Avoid negative pressure gradients at the rear part.
- The underbody should be smooth and continuous.
- Avoid sharp angles (except to compensate cross-wind instability).
- The front start at a low point and goes up in a smooth line.
- The side pontoons for cooling have is designed for minimizing drag.

According to the characteristics, the goals of this project are:

- Make an efficient aerodynamic design, maximizing the downforce without increasing the drag force excessively.
- Make a cheap design, so that, the team can send it to manufacture.
- Make a simple design, easy to change or remove parts quickly during the competition in case of car reparations.

Chapter 2

Aerodynamic elements

2.1 History

In 1921, Edmund Rumpler, a German inventor, created the Rumpler-Tropfenaut that was based on the teardrop, one of the less drag shapes that naturally appear. The car has a drag coefficient of just 0.27 [16]. In USA, the Chrysler Airflow was one of the first cars designed taking into account aerodynamics and its shape can be seen in figure 2.1. [17]

Figure 2.1: Chrisler Airflow car

Figure 2.2: Mercedes 1954 F1

The aerodynamic shapes where develop in competitions. Many examples can be mention, especially for fast circuits, like the Mercedes Formula 1 1954 on figure 2.2.

The bodywork firstly were made with veneer, after a while, they decide to made it in aluminum, light alloys and even magnesium alloys. Currently it is used polyester resins reinforced with glass fiber. They has excellent mechanical properties and has the advantage of light weight. In the case of Formula 1 cars it is used carbon fiber. In this situation, the weight reduction with respect to a metal sheet can be up to 70% [18].

For consumer market, companies like Citroën, Lotus and Porsche developed some streamlined designs, but these models were applied mainly to high-performance sports cars and not vehicles for common drivers. Today, nearly all cars are designed taking into account aerodynamics to reduce consumption and better performance to meet environmental regulations. [19]

2.2 Nose and wings

In a first approach, a higher nose produce less downforce because it pushes less quantity of air to the upper part of the car. This situation allows the air go straight under the nose and car body instead of bend and go around the car as can be seen in 2.4. [20]

Figure 2.3: Red Bull RB1

The front wing can generate downforce to be applied in the front axis. All air that goes under the nose is then guide under the car or split and conduct it to the sidepods. As much air is introduced under the floor and as fast it can exit out of the diffuser, more downforce will be generated. [21]

Figure 2.4: McLaren MP4 comparative

The nose cone has to be design to maximize aerodynamic efficiency and it has to comply the requirements of strength and the measurements established by the organization of the event, in figure 2.4, it is showed a comparison between two consecutive F1 cars. Also it has to absorb a certain quantity of energy in case of frontal collision, and support the front wing. In figure 2.5 there is model to analyze the frontal impact by computer simulation.

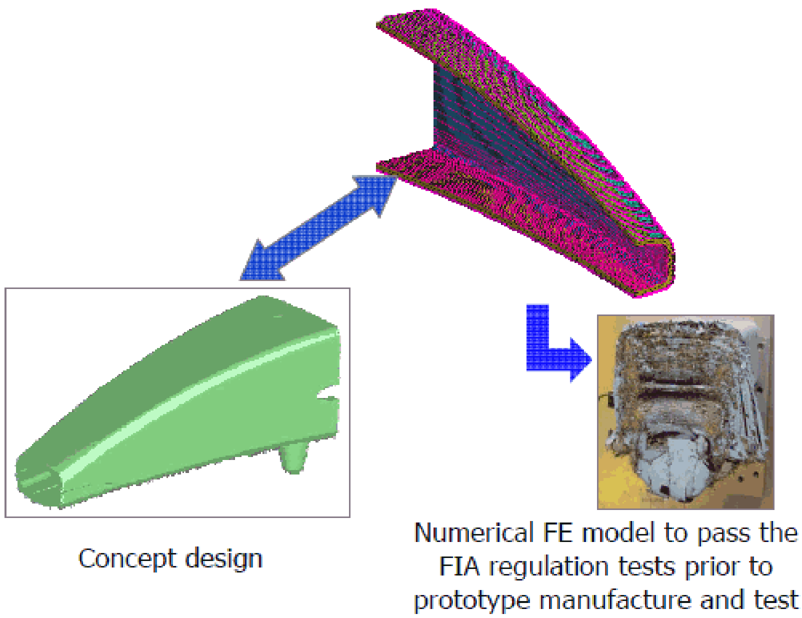


Figure 2.5: Finite Element Method nose cone

In a race car, the wing is the element that works as an airplane but in an inverse way, so the air though it push the vehicle to the ground. As can be seen in equation 2.1 as bigger is the normal force applied, bigger is the friction force with the ground, and then, bigger the traction and grip.

The used of small and multiple elements facilitates their adjustment to regulate the aerodynamic force during the race. Some wings have Gurney flaps and lateral drifts that avoid detachment of the boundary layer converting a laminar flow into turbulent flow. A turbulent flow is worse than a laminar flow because it has a bigger boundary layer, but this is better in terms of drag and lift than having detachment of the flow. [22]

$$F_f = \mu N \tag{2.1}$$

Where:
μ is the adherent coefficient.
N is the normal force.

The ideal working conditions do not exist in the real world because the car goes through a track while the wheel turns, the suspension acts and maybe there could be another car in front of ours and it creates what is called dirty air, that means the flow is not laminar. A good design has to reduce the effect of this real situations.

A wing is design with a plane and a curve face or, two curve faces being one of them larger in length that means, the air has to go quickly in one part than in the other one. According to the Bernoulli principle, it generates a lower pressure in the face that the fluid goes quickly. Then, there is a resultant force due to the difference of pressures, as shows equation 2.2.

$$F_l = (P_{down} - P_{up}) \cdot A \tag{2.2}$$

Where:
P_{down} is the pressure in the lower surface.
P_{up} is the pressure in the upper surface.
A is the area of the wing.

Figure 2.6: Front wing example

A race car, can have wings with more than one elements or flaps as in figure 2.6, it will depend on the rules of the competition.

The point where theoretically the forces are acting is called center of pressure and there is not any resultant moment on it. The ratio between lift and drag is used to measure the aerodynamic efficiency.

In the case of the wings, the aerodynamic force grow up as the attack angle increase until, it reach a limit value where that value goes down, in this situation, the flow separates the surfaces decreasing the downforce and increasing the drag force.

The height and cross section in the nose is very important because they want a minimal cross section in the front of the car for reducing the drag force, which is proportional to the cross sectional area, and distribute the air flow to the side or under the car, increasing stability and grip. [1]

The nose cone and the front wings have elements known as attachment pylons. This elements, can be design taking into account the aerodynamic effect and how they perform in the drag and lift forces.

2.3 Endplates and air intakes

The engine air intake of a Formula car is located above and behind the driver's head to get undisturbed and clean air that the engine can use, with the less pressure losses showing in 2.8.

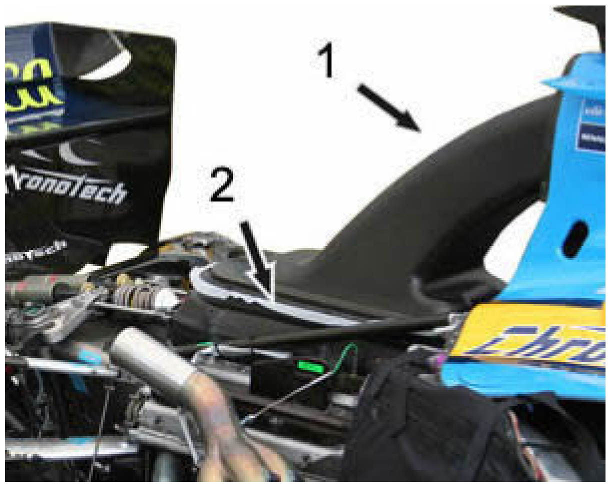


Figure 2.7: Renault F1 intake

Inside the air intake there is an expansion chamber that slow down the air and prepare the flow for its passage into the engine inlet manifold as can be seen in 2.7. The air intake is positioned away from sources of heat, such as the track and radiators, to minimize the air temperature what allows more efficiency in the engine. [23]

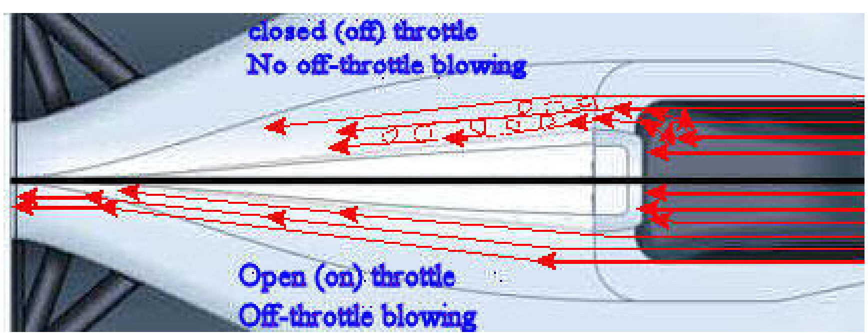


Figure 2.8: Airflow lines in a formula car

In a thermal combustion engine is very important the refrigeration for its operation. The oil is pumped through a radiator to cool down before completing another cycle through the engine. This oil is also use to lubricate the mechanical parts of the engine.

The main purpose of formula cars side pontoons is to provide enough airflow for cooling systems, that place the radiators in the sides and the lower part of the car, without sacrificing aerodynamic efficiency and vehicle safety. The shape depends of the properties of the air flow and is determinated by the front wing of the car.

2.4 Flat bottom and diffuser

In any race car, there is a difference between the upper flow coming from above the driver's head and the down flow coming from the bottom of the car. If the car has an irregular bottom it contributes to increase the boundary layer up to the ground, blocking the flow. Because of this, increasing the drag force and the turbulence behind the car. This blocked flow creates high pressure zones in the lower part that is the opposite what is desirable to get a negative lift force or more grip.

To analyze the ground effect it is need to consider:

- The height of the car with respect the ground. The less higher is the car, the more downforce is get.
- The irregularity of the bottom. The more regular the shape is, the more downforce is achieve.
- The bottom shape. The smoother the shape is, the more downforce is get.

To reduce the drag and increase the downforce, it can be design the bottom like a nozzle, smoothing the air inlet and delaying the growth of the boundary layer, also increasing the speed of the air and generating a low pressure at the exit, that means more grip.

The variation of drag and lift versus distance with respect the ground for a wing is show with the figure 2.9. Being h the height and c the length of the wing.

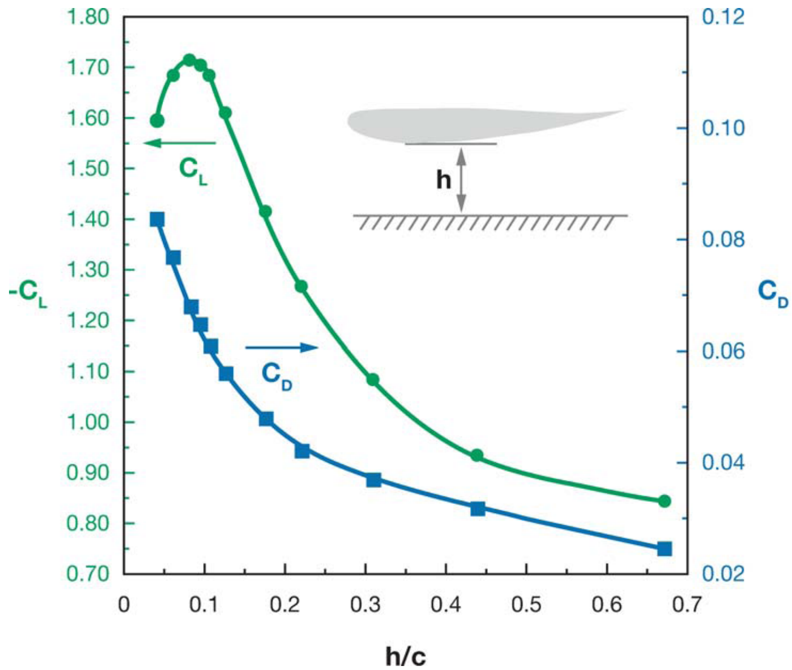


Figure 2.9: Ground effect graph

After choosing the form and height of the car bottom, it can be done two alternatives:

- Block the air flow that goes under the car.
- Allows the flow go under it but in controlled conditions.

For this case it was considered to allow the flow, but reducing the amount of it due to a lower nose cone and low height.

The diffuser is an aerodynamic element that is located in the rear part of the car and in its bottom. This piece allows open the bottom flow to the upper flow and the influence of the low pressure zone created by the turbulence. [2]

The underbody shape causes an increment of air volume while the car is moving, so the air has to accelerate to fill that volume, decreasing the pressure and increasing the downforce. After the diffuser, the air gets slower in a smooth path to join the air of the rear part getting the less drag force as possible. It can be seen in figures 2.10 and 2.11

This path, facilitates the transition between the air of the bottom and the freestream airflow of the upper part. It decelerates the underbody flow reducing flow separation and drag as possible.

Also, the diffuser, accelerates the fluid before the fluid gets inside it to generate more downforce. It uses the Bernoulli's principle, a simplification of the Navier-Stokes equations, for a non-compressible and steady flow, such that, the pressure decreases while the velocity increases. It can be also consider, the injection of the exhaust system into the diffuser that can generate more downforce.

Fast gases coming from the exhaust system helps to move the air of the diffuser faster than without it, which helps to reduce the pressure at the underbody. However, this influence depending of the engine revolutions per minute, and when the driver lifts off the throttle, the exhaust flow is reduced, or even disappear, and means less downforce.

Clean air, which is also known as a non perturbated flow under the car delayed flow separation in the diffuser. In addition the rear wing can helps the diffuser, in a way, such as closer it mounted, more help. [24]

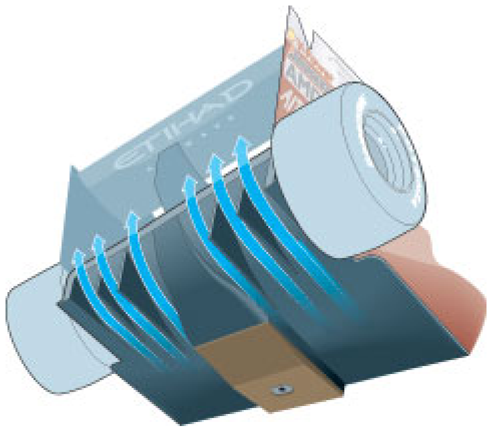


Figure 2.10: Single diffuser sketch

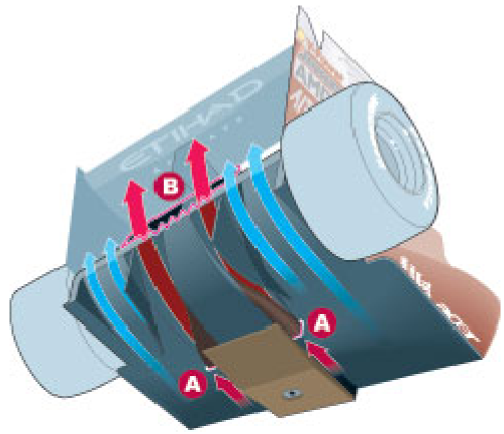


Figure 2.11: Double diffuser sketch

Chapter 3

Aerodynamic theory

3.1 Introduction

A car that moves, crosses an air-wall every little position that it advances. At high speeds, or, on windy days, air resistance has a great effect on the way that a car moves, handles and increases fuel consumption.

Aerodynamics is the study of forces and the motion of particles or objects through a fluid, in this case, the air. Since long time ago, all the machines that moves have been designed with aerodynamics in mind, and companies have develop innovations that make cutting through that air-wall easier and comfortable.

Having a car designed taking into account airflow means it can accelerate easily and can achieve better fuel economy because the power devices have to work less.

For instance, rounded shapes on the edges of the vehicle are useful to channel air in a way that it flows around the car with the least resistance possible, that means, the turbulence and the boundary layer separation are minimize. Some race or sports cars even have parts that move air smoothly across the underside of the car. For this competition, this kind of elements are forbidden.

When an object is moving through the atmosphere, it displaces the air that is in front of it to the back side. This object, is subjected to gravity and some forces, like drag and lift. Drag force increases with velocity in a square relation, as fast the object travels, more drag it experiences. Drag has a direct effect on the acceleration like is shown in figure 3.1 [25]

$$Acceleration = \frac{Thrust - Drag}{Mass} \tag{3.1}$$

3.2 Concept of fluid

A fluid deforms during the application of shear stress, it adapts to the shape of their container, which is opposed to solids, it has not resistance forces. Compared with a solid, it is said that the solid molecules are strongly bounded while in a liquid they can move freely and separated themselves without offering much resistance.

3.3 Ideal fluid

A precise study of all variables in a real problem can be complex, because of this, it is needed to make some simplifications that do not change the solution so much, at least, in the same order of magnitude they are working and do the calculations much easier for us and also for the computers. [3]

Therefore it is studied the fluids called ideals. Usually these is different from the real fluid in the effect of viscous forces. In other words, fluids are considered ideal that have no internal friction between fluid particles.

3.3.1 Gases and liquids

There are two main types of fluids that are liquids and gases. Liquids have cohesive forces between the molecules, which cause the fluid to maintain its constant volume. In the other hand, gases expand to occupy the maximum volume.

3.3.2 Compressibility

All fluids are compressible, but when it is talked about non-compressible fluids is refered to this property which is negligible. The gases are highly compressible fluids, while liquids are generally considered non-compressible.

3.3.3 Viscosity

Viscosity is the opposition of a fluid to shear stresses. If it is considered a fluid consists in a series of layers, the viscosity represent the friction between those layers when a force is applied.

3.3.4 Convection

Convection is the phenomenon where the air moves because it has different temperatures. Warm air is less dense than cold air, so it tends to go up. This movement causes heat transfer from lower layers to the upper layers.

3.4 Velocity field

Each particle of a fluid have different speed and acceleration. Then, the velocity field have to be describe as three dimensional vectors in terms of speed and acceleration. The velocity vector is defined as equation 3.2. However the velocity field is as equation 3.3

$$\vec{q} = \frac{dx}{dt}\vec{i} + \frac{dy}{dt}\vec{j} + \frac{dz}{dt}\vec{k} \tag{3.2}$$

$$V(x, y, z, t) = u(x, y, z, t)\vec{i} + v(x, y, z, t)\vec{j} + w(x, y, z, t)\vec{k} \tag{3.3}$$

Where:
x, y, z are orthogonal coordinates.
t is the time.
i, j, k are orthogonal vectors.

The are two methods to describe the movement of particles in a continuum media:

- Lagrangian: the coordinates of the particles are in function of the time, so for a given time, the position is represented as equations 3.4, 3.5 and 3.6

$$x = (x_0, y_0, z_0, t) \tag{3.4}$$
$$y = (x_0, y_0, z_0, t) \tag{3.5}$$
$$z = (x_0, y_0, z_0, t) \tag{3.6}$$

Being the x_0 y_0 and z_0 the initial coordinates in a given time t_0.

This method is uncomfortable to work with it, because there are many parameters in the equations. Also it is not interesting to follow the path of a single particle but the behavior of the whole fluid in the media.

- Euler method: it allows to know for a certain point in the space, the description of the characteristics of the flow. In this method the speed is as a function of time, it can be checked in equations 3.7 and 3.8

$$u_{i+1} = u_i + \left(\frac{du}{dt}\right)_i \Delta t \tag{3.7}$$
$$u(0) = 0 \tag{3.8}$$

3.5 Trajectories and current lines

A current line is an imaginary curve that connect points in the space and the particles that are in that line, they have velocity vectors tangent to the curve, it is showed in figure 3.1.

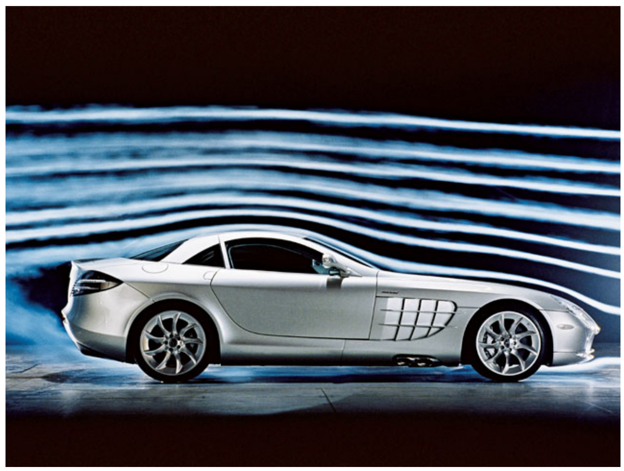


Figure 3.1: Current lines in a car

In an steady flow, the current lines are fixed with respect to the reference system and they coincide with the trajectory of the particles. For a two dimensional current line it can be deduced equation 3.9

$$\frac{dx}{u} = \frac{dy}{v} \tag{3.9}$$

And in general on three dimensions equations 3.10, 3.11 and 3.12.

$$vdx = udy \tag{3.10}$$

$$wdx = udz \tag{3.11}$$

$$udy = vdz \tag{3.12}$$

3.6 Continuity equation

The continuity equation explains the conservation of the mass between the input and the output, because the air mass that enters in the front of the car has to go out exactly the same quantity. The mass does not create or destroy it has to conservate. [3]

$$\frac{\delta}{\delta t}\int(\rho dV) = \int \rho q_n dA_i - \int \rho q_n dA_e \tag{3.13}$$

Where:
ρ is density
t is the time
V is the volume
A is the area
q_n is the velocity vector

If it is a constant velocity, the equation 3.13 is equal to zero.

3.7 Boundary layer. Laminar and turbulent flow

A fluid flow have two different behavior depending of the type of fluid, the geometry and the speed. For this project it was assumed that the fluid is air, and its properties does not change due to temperature, pressure and humidity. There is an adimensional parameter that relates the inertia forces with respect viscosity, this parameter is called Reynolds, and is defined in equation 3.14

$$Re = \frac{\rho v L_c}{\mu} \tag{3.14}$$

Where:
v is the speed.
L_c is the characteristic length.
μ is the viscosity.

This parameter predicts the behavior of the fluid: laminar for a low values and turbulent for high values.

3.7.1 Laminar flow

The laminar flow follows a smooth line, the fluid moves in parallel sheets without mixing each others. In the laminar flow the tangential efforts tend to be high, even for small velocity gradients, but de inertial force has low importance respect to viscous force.

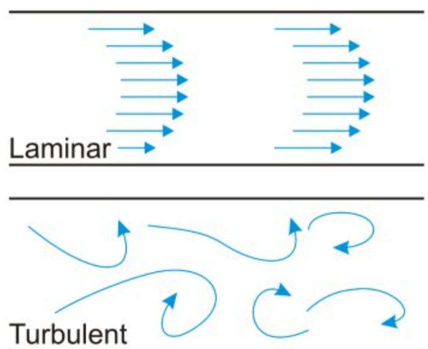


Figure 3.2: Laminar and turbulent description

3.7.2 Turbulent flow

The turbulent flow is a chaotic movement of the fluid particles and it creates vortex. It has high momentum convection but low diffusion, and quick exchange of pressure and velocity in time and position. In figure 3.2 it can be seen a comparison between the laminar and turbulent flow.

A fluid that contacts a wall has no velocity, and as result of this, the gradients of the speed and shear forces have maximum values in the border or parts touching walls and they decrease into the fluid.

This thin layer, is called boundary layer where the speed gradients decrease quickly and the shear effort is also low. Inside it, the viscous forces are much bigger than inertial ones. But out of the layer, the flow has a domination of the inertial efforts and the pressure gradients, as a result of this, the current lines, corresponds to a speed potential, as can be seen in figure 3.3.

The limit of this layer is consider when the particles get the 99 % of the speed of the flow and the thickness could be from a few molecules to few millimeters, depending of speed, viscosity or geometry.

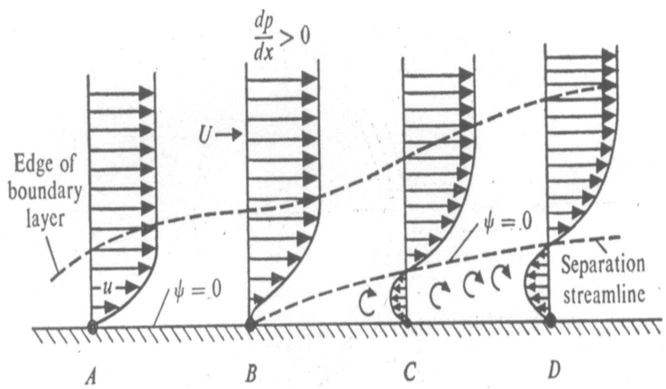


Figure 3.3: Boundary layer description

A fluid that moves with respect a rigid wall produce a force to it, it happens because of:

- The shear forces due to the viscosity and the speed gradients in the border surface have tangential forces.
- The pressure produces normal force at the wall.

For a body which is in a fluid, the vectorial sum of the forces, tangential and normal, integrated in the whole surface gives a resultant force vector. The component of this force in the direction of the relative speed is the drag and the normal component is the lift. [26]

The drag and lift coefficients are defined as:

$$C_d = \frac{F_d}{\frac{1}{2}\rho A V^2} \tag{3.15}$$

$$C_l = \frac{F_l}{\frac{1}{2}\rho A V^2} \tag{3.16}$$

Where:
F_d is the drag force.
F_l is the lift force.
ρ is air density.
A is the frontal area.
V is the velocity.

3.8 The Bernoulli effect

One physical effect involved in downforce generation is called the Bernoulli effect. This means that if a fluid flows around an object at different speeds, the slower part of the fluid generates more pressure than the faster, then the object will be forced toward the faster moving fluid, as can be seen in figure 3.4. [24]

In car aerodynamics it is needed to get high air speed or low pressure air to go under the car creating negative lift, for facilitating the car have good grip when it runs. The faster the car goes, more downforce is get and more grip will be obtain, pushing the car to the track.

As more grip it has, more traction, especially going through the corners, because as it double the speed, it is get four times downforce and drag. [27]

A Formula 1 car can produce enough downforce to drive upside down, so it can produce more negative lift than the weight of the car at fast speed.

Race teams have dedicated engineers striving continually to create more effective designs because this can make the difference between first and last place. [28]

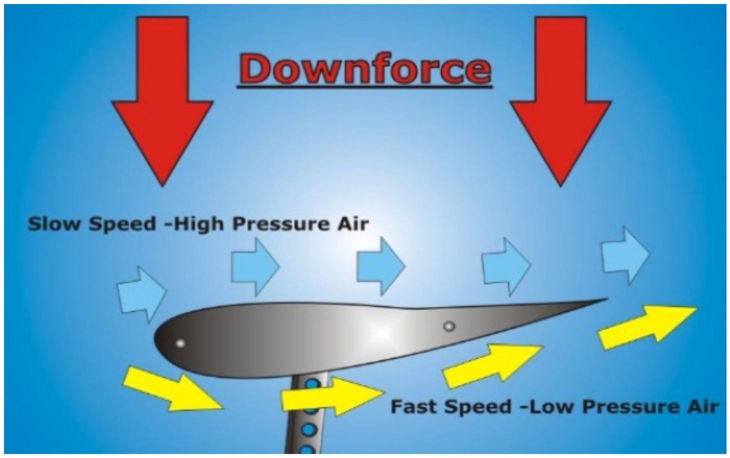


Figure 3.4: Downforce

3.9 The drag coefficient

The aerodynamic efficiency of a car are measured using the drag coefficient of the vehicle. As low C_d is, more aerodynamic a car is, and easier it can move through the wall of air, shown in figure 3.5. [24]

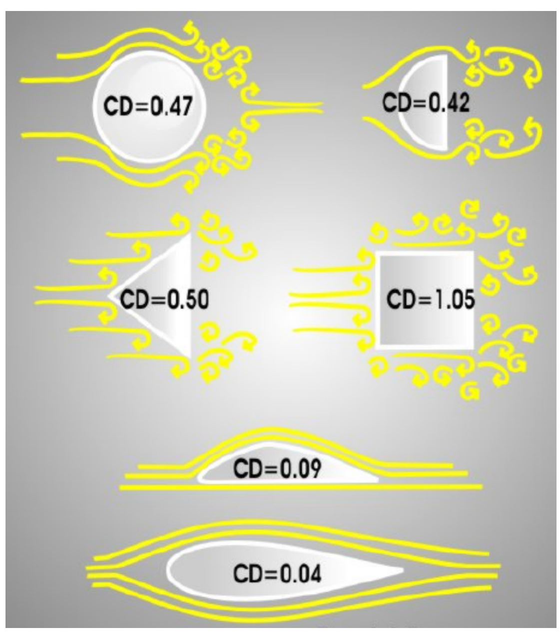


Figure 3.5: Drag coefficients shapes

Today, most cars have a C_d of about 0.30. But SUVs, are larger, so they have a C_d from 0.30 to 0.40 or more. In figure 3.6 it can be seen the shape that is less aerodynamic. [29]

Figure 3.6: SUV shape

Figure 3.7: Mini stream lines

The Toyota Prius hybrid has an aerodynamic shape because its C_d of 0.26 helps it achieve very high mileage, shown in 3.8 . In fact, reducing the C_d of a car in 0.01 it result as 0.09 kilometers per liter increase in fuel economy. [30]

Figure 3.8: Toyota Prius

Some cars have active car aerodynamics, for instance a wing moves depending of the speed, or appears an aerobrake under heavy braking acting to stabilize rear traction like the Bugatti Veyron, which top speed is 407 km/h. It can be seen in 3.9.

Figure 3.9: Bugatti Veyron

The bigger downforce quantity is generated through underbody diffuser design, which produces low drag levels. The most speed vehicle is a rocket, shown in figure 3.10, the design is predominately engineered to cut through the air, so it minimize drag levels. Some cars are design, taking this into account, but are not appropriated for everyday cars.

Figure 3.10: Rocket car

3.10 Aerodynamic add-ons

A typical Formula 1 car has a C_d of about 0.70. This is because are built to generate as much downforce as possible. The car achieve speeds that with their light weight, this car actually begin to experience lift like an airplane, and it could mean a crash. For this reason, downforce is maximized to keep the car on the ground at high speeds. In figure 3.11 it is shown a comparation between drag coefficient of different bodies.

Formula cars get this using wings mounted on the front and rear of the vehicle, pressing the car to the ground and allowing to achieve higher cornering speed.

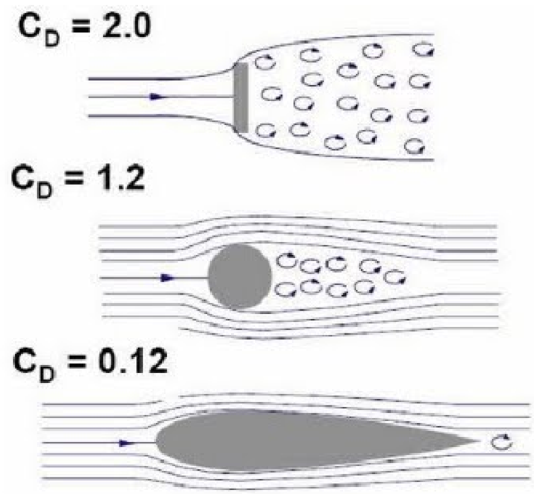


Figure 3.11: Drag coefficient comparative

Chapter 4

Computer Fluid Dynamics

4.1 Introduction

Computer Fluid Dynamics, or shortly, CFD, study the behavior of fluids using computer simulations. Its objective is to find a solution about fluid equations, dividing the space in small elements called mesh, solving in each one, the equations and relating the solution in each cell with the neighbours. [4]

CFD is a very useful tool because it allows to the designer:

- Reduce time and cost of designs, including radical changes. The design can be tested by simulation and get an approximated behavior of it before manufactured.
- Possibility to analyze conditions that in the real world could happen but are difficult to simulate, such as supersonic speeds, extreme temperatures or tornadoes.
- Capability to study danger substances.
- The resolution is only limited by the computer time.
- Without CFD, the process to design complex geometries is trial and error.
- CFD provides a tool to understand the fluid behavior in all parts of the car body that could not be able to achieve using only the wind tunnel as can be seen in figure 4.1.

- Useful tool but it has to be complemented with wind tunnel and some physical test because this numerical simulations have an error about a 10%, then for high-performance applications is not good to trust completely in it.

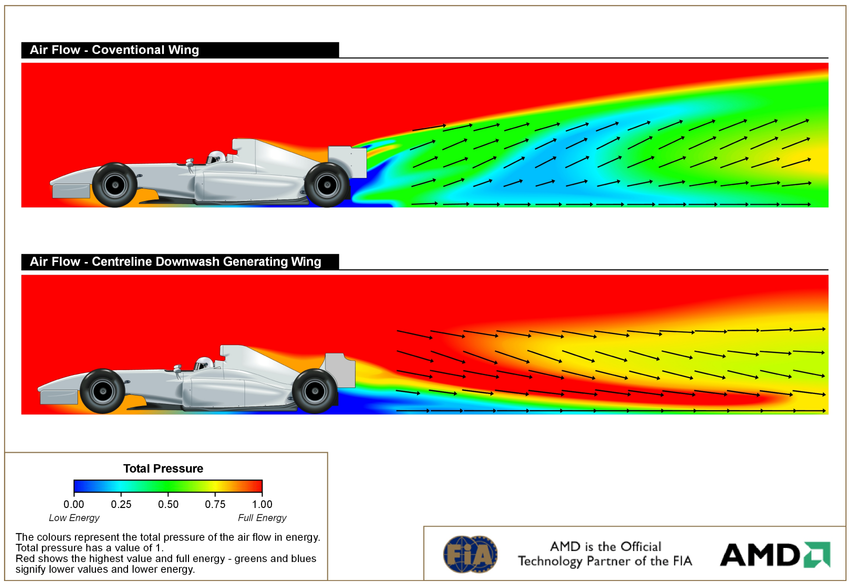


Figure 4.1: CFD in F1

Computers are used to simulate the interaction of fluids with complex shapes in engineering projects. Even using simplifications in the equations and a good computers, it can only obtain approximate results in most of the cases. The verification of the computations is normally compare with scale models in wind tunnels.

The method creates a discretized region of space doing what is called a spatial grid, with means the control volume space in very small regions. The computer solve the equations using iterations and it stop when the differences between one iteration and the next one is below a value that it is imposed. When it happend, the solution had converge. [5]

4.2 Fluid analysis

The laws to know the movement of a particle in a fluid are known as Navier-Stokes equations. The boundary conditions and initial conditions are also important to solve the problem. It is different one case to the other for a specific problem.

The initial conditions defined the fluid at null time in its starting point. For doing the calculations, it is need to know all the variables in this point for to get a good result. As sometimes it is not possible, it is used to assume that in an stationary state the boundary conditions are constants.

4.3 Models

There are many different models to find solutions, the application of each one, depends of the problem and the simulation that it is going to solve because each one fit better in a specific situation.

The models are:

- Spalart-Allmaras with one equation.
- The k-ϵ standard with two equations.
- SST k-ω with two equations.
- Transition model k-kl-ω with three equations.
- SST of transition with four equations.

Also, there are different approximations to solve the boundary layer close to the surfaces:

- The integral method that solve equations of the boundary layer is complicated to implement in non-structurated meshes because it depends of the variables to integrate.
- In the boundary layer it is need a fine mesh enough to solve the zone affected by the viscosity.

The turbulence appears when the inertia forces are bigger than the viscous, so the Reynolds number reach a certain value that some non-stationary phenomena are produce. This turbulent fluctuations make the fluid more diffusive increasing the mass transportation and the exchange of the momentum and energy.

Turbulence physically shows the generation of vortices according to Kolmogorov's theory from 1941 that says the interaction extract energy from the flow and transforms it into internal energy to deform the fluid particles. This is the reason that the turbulent flow dissipates more energy than laminar flow. The interaction of the vortices causes vortices subdivision increasingly smaller, so that different scales coexist.

It is possible directly resolve all spatial and temporal scales of the flow turbulent, not averaged, using the Navier-Stokes equations. This is called DNS, Direct Numerical Simulation, and the discretization of the fluid must be of the same order of magnitude of swirls microscale. Obviously this has a high computational cost, and it is applicable to studies of small dimensions, especially in the field of research.

For applications in the engineering, Eulerian models typically work with average values and incorporate a swirl pattern that reproduces the effects of turbulence in the flow. In engineering is usually sufficient to study the effects of the mean flow, making a statistical approach for a period of time much larger than the characteristic period of the turbulent fluctuations.

4.4 K - ϵ model

There were developed many k - ϵ models, the first one were done by Launder Jones in 1972 and it is known as the standard k - ϵ whose coefficients were adjusted shortly by Launder and Sharma in 1974. The k - ϵ model is the most known and used in virtually all commercial programs to study fluid, for instance, SolidWorks.

The k - ϵ model is a semi-empirical model based on the transport equations for turbulent kinetic energy (k) and the dissipation of the turbulent kinetic energy (ϵ). In deriving the model, it is assumed that the flow is fully turbulent and that effects of molecular viscosity are negligible.

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{4.1}$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{\partial}{\partial x_i}\left[-\left(\frac{P}{\rho} + \frac{2k}{3}\right)\delta_{ij} + 2\left(v + v_T\right)S_{ij}\right] \tag{4.2}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v\frac{v_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \Pi - \epsilon \tag{4.3}$$

$$\frac{\partial \epsilon}{\partial t} + U_j \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v\frac{v_T}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + c_{\epsilon 1}\Pi\frac{\epsilon}{k} - c_{\epsilon 2}\frac{\epsilon^2}{k} \tag{4.4}$$

where:
Equation 4.1 is the equation of mass conservation.
Equation 4.2 is the momentum equation.
Equation 4.3 and 4.4 are the transport k and ϵ model equations.

And:
U_i is the velocity.
P is the pressure.
$v_T = \frac{c_\mu k^2}{\epsilon}$ is the viscosity.
S_{ij} is the tensor of the deformation flow rate.
$Pi = 2v_T S_{ij} S_{ij} \geq 0$ is the kinetic energy production rate.
The last two terms in 4.4 are called production and destruction of ϵ.
$c_{\epsilon 1}, c_{\epsilon 2}, c_\mu, \sigma_k, \sigma_\epsilon$ are numerical coefficients to adjust the model.

Chapter 5

Design

5.1 Introduction

The bodywork design was made with the commercial software SolidWorks that the team got by sponsorship. This program has all the tools to do the complete car design like Computer Fluid Dynamics tool, Finite Element Method tool, motion simulator or electrical circuit simulator. All the calculations have an approximately error of a 10%.

When it is designed a new model, it should combine a number of factors including comfort, aerodynamics and security, aimed at obtaining a product that offers significant power with fuel economy. As an example in a sport car habitability is sacrificed in favor of aesthetics and aerodynamics, in a minivan what prevails is the interior layout background instead of aerodynamics. Trying to achieve these objectives are used different strategies:

- Optimize organizational tasks of all divisions involved in the development of the new model allowing a quickly detection of any problem present.
- Application of new concepts and new technologies.
- Capacity for innovation. The strength of a team resides in its ability to innovate faster than competitors.

The idea of security is in the forefront when designing a body not only aesthetics. It study the deformation of their creations.

A typical sequence for a Formula Student new car is developed in the following phases:

- First scketches.
- Design.
- Creating models.
- Building.
- Tests.
- Competition.

5.2 First sketches

In this phase, the first designs are made from hand drawings. Then, the technical director working with the team determine the dimensions of the vehicle.

For the initial calculation of the exterior body measures usually take into account:

- Aerodynamic requirements.
- Cockpit ergonomics.
- Position and size of the fuel tank.
- Space requirement for wheels.
- Size and arrangement of the suspension.
- Type of position of mechanical parts: engine, radiator, change.

5.3 Control volume

A control volume is an arbitrary space where an object is placed. This space is delimited by a close surface that it can be real o virtual and this space keep its shape and volume. This means, that in case of a CFD the control volume is fixed and the air particles moves across it, so some particles enter in it, and some go out, but the mass energy has to be the same.

For doing the simulations, it was considered a control volume, as can be seen in figure 5.1, enough to analyze the trail and its behavior with the car and side dimensions, that not influence in our calculations. This control volume was optimize after some calculations balancing the computation time and the residuals of the forces that are calculating in the iterations, whose measures are in table 5.1. The X axis is placed along the car, the Y axis is placed vertically and the Z axis is the lateral one.

Dimension	Value [m]
X_{min}	-2.5
X_{max}	15
Y_{min}	-1
Y_{max}	2
Z_{min}	-3
Z_{max}	3

Table 5.1: Control volume dimensions

All the calculations are done for 90 km/h that is just a hypothesis of the top speed of the car during the event and taking into account the power and the length of the track.

The car of figure 5.1 is placed in the center because of in the real case, the relative velocity of the ground with respect the air is zero, and this effect is not possible to simulate with the software. The only possibility is putting a static wall close to the car bottom but it will have a relative velocity respect to the air and it produces some results that are not corresponding to the reality.

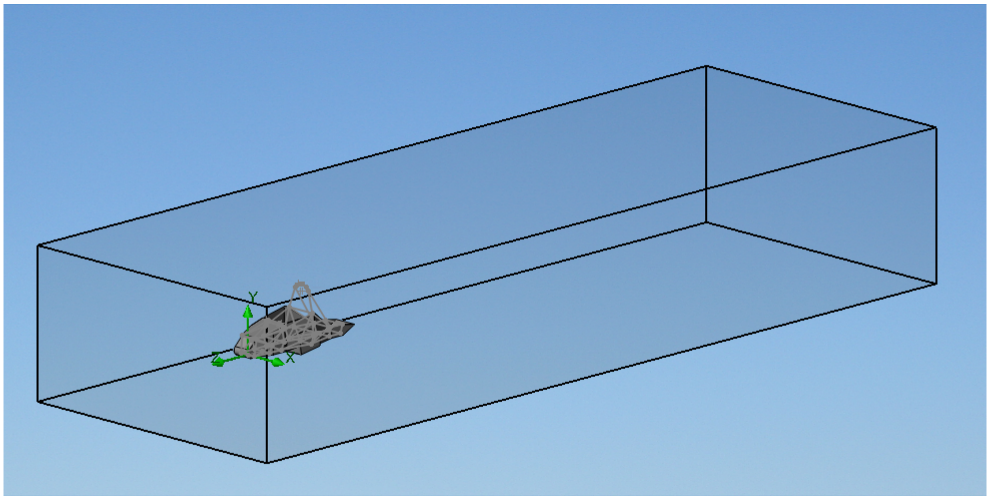


Figure 5.1: Control volume

Then, if the car is in the center and far from the limits of the control volume the results have good quality and translating them to reality, they just improve the car behavior due to the Venturi effect of the car passing.

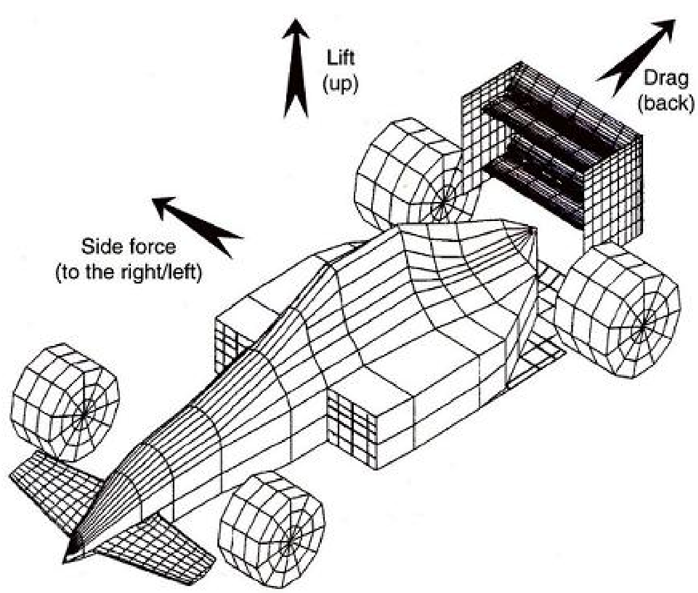


Figure 5.2: Car forces

The figure 5.2 shows the forces that a car experimented. These forces are obtained with the simulations. The forces are drag, lift and lateral. Drag means the air resistance to movement. Lift, which is the vertical force experienced by the car due to Bernoulli's principle, this force is important that every effort is negative, i.e. towards down. In the case of the lateral, it is suitable to be zero and using the wings to control understeering and oversteering.

5.4 Ahmed body

In the early 1980s, the Ahmed body was tested. It consists a simpler model car shape that allows to research in the behavior of new turbulence models developed. The Ahmed body is a round front part and a rectangular box, which connects the front part and the rear part.

A spite of neglecting a number of variables that a real car has, like rotation of the wheels, roughness of the bottom or aerodynamic elements, the Ahmed body generates the most important features of flow around a car, for instance: flow around the nose, relatively uniform flow around the middle and flow separation and wake generation at the rear. In figure 5.3 it is shown an usual Ahmed body used in research.

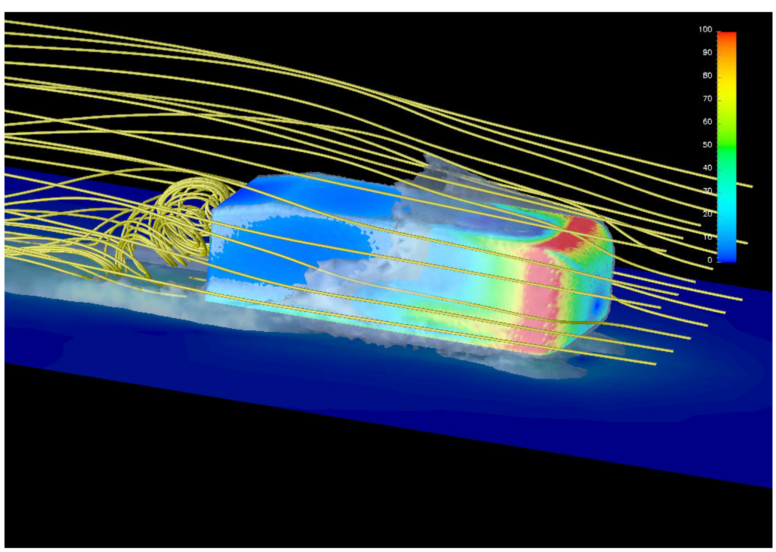


Figure 5.3: Usual Ahmed body

The principal goal of studying such a simplified car shape is to understand the flow processes involved in the drag forces. If it is understood how drag is generated, it should be able to design a car to minimize drag and therefore decrease fuel consumption and improve performance.

The rear part of the vehicle provides the principal contribution to pressure drag and it is critical determining how the wake flow take placed and hence the drag experienced by the vehicle.

It was suggested that the low base pressure was caused by a vortex which it was too strong. It was the result of large wall shear stress being predicted by the k - ε model. The linear k - ε model predicts the turbulent kinetic energy at a stagnation point which would encourage the flow to remain attached. This indicate a model which responds more accurately to adverse pressure gradients is able to account for the complex behavior on the rear part of the Ahmed body.

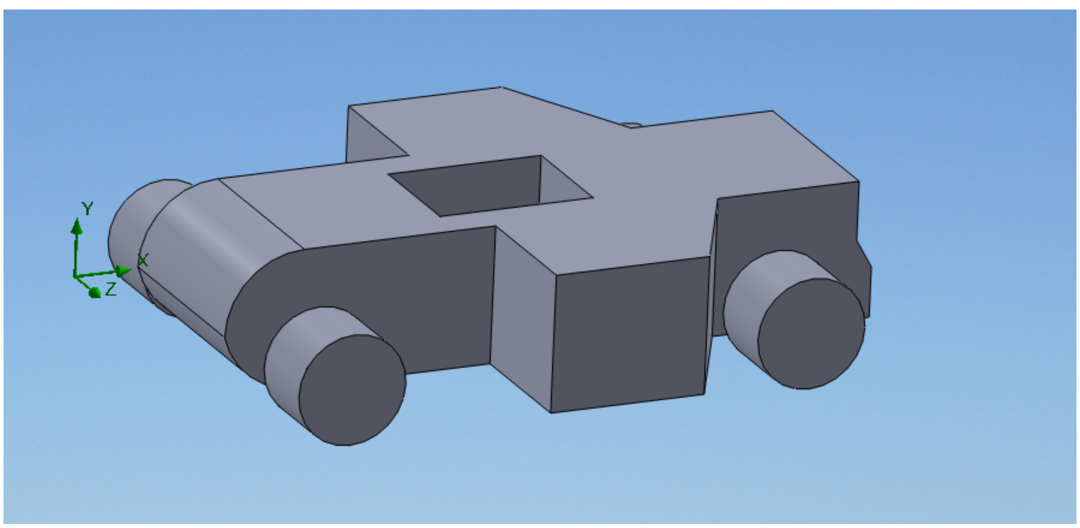


Figure 5.4: Ahmed body model

Then, to start analyzing parameters for the CFD study, such as the mesh, the influence of gravity and the boundary conditions. It was constructed an Ahmed body with a formula car shape. It has a round nose, square body including a hole where the driver is placed. It also takes into account the non-covered wheels, the lateral pontoons and a diffuser in the rear part.

5.5 First designs

After analyzing the influence of the different parts of the Ahmed body in the drag and lift forces. It was constructed a bodywork using the surface tool creating different sketches at different sections of along the formula car, for instance where the nose cone start, after that where the brake deposits are or where the copkit begins.

This model can be seen in figure 5.5, it was complicated to change the geometrical parameters without having errors with the software. Also, it was not possible to draw a diffuser with a surface mode and the nose cone was a problem source.

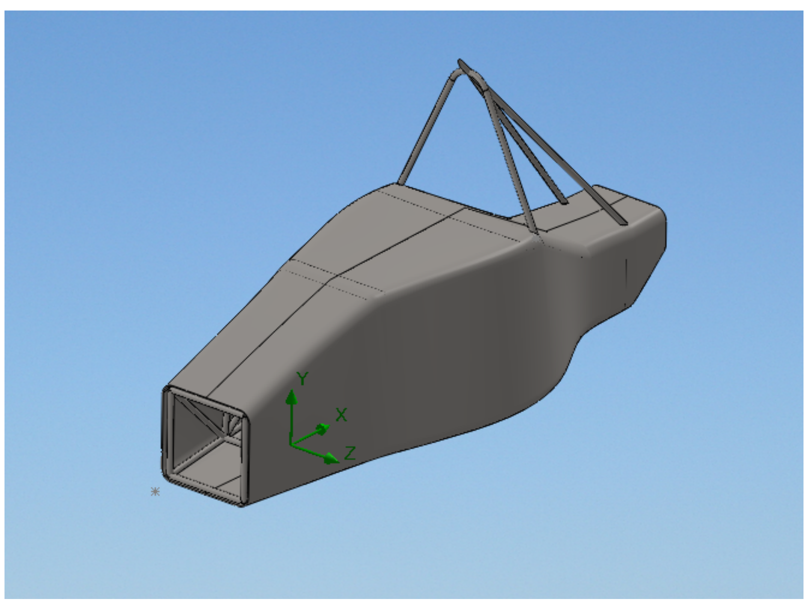


Figure 5.5: Aerodynamic design using surfaces

Because of this, the final design was done with the solid mode using a empty solid with a constant thickness, that allow to change the geometrical parameters to analyze parametrically.

5.6 Nose

The nose is the first part of the car that contact the air wall, then, it is important how this element divides the air that goes behind and how is its behavior. In the down part, it is critical how much air go under the car to maximize the ground effect. In the upper part, the fluid has to delayed as much as possible the detachment of the boundary layer.

In the design it was analyzed the influence of the height of the nose in terms of drag and lift forces. In figure 5.6 it can be seen the geometrical parameters that there were considered to study the nose cone. The NLU and NHU parameter are related with the upper slope. Also for NH and NL dimensions, but for the lower slope. The height is the NH plus the radius of the cone, that is NF.

It has been considered only the first part of the nose, which takes from the initial point of the cone to the first tubes. The second part it has been designed closer to the tube as much as possible for reducing the frontal area.

The possibility of doing a smooth nose with the same angle is discarded due to the rules specified a maximum length so it can not be possible.

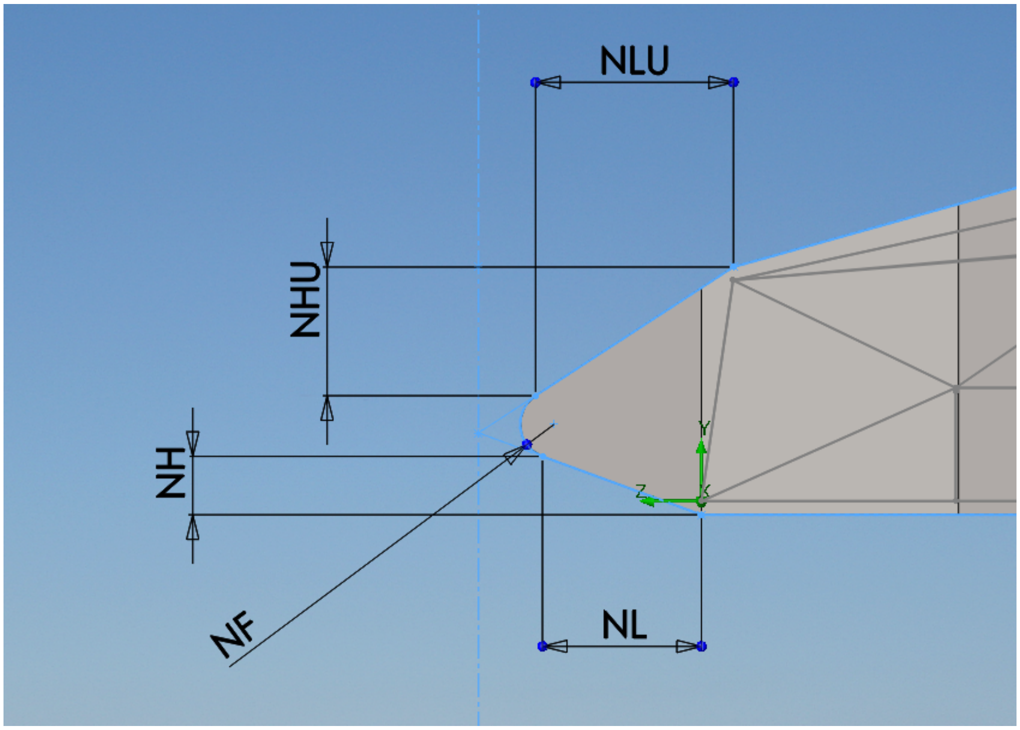


Figure 5.6: Nose parameters

As can be seen in the graphic 5.7 there are two points that are good to take into account, because the lift is low and also the drag. In figure 5.8 it is shown that the best option is less radius cone but taking into acccount the limitations of rules, it was chosen 100 mm of radius. The characteristics of the these nose are resume in table 5.2 and then, in table 5.3 the forces of each design.

Model	Height	Radius	Width	Bottom slope	Upper slope
Nose A	0	100	394	0	43.75 °
Nose B	160	100	394	16.18 °	28.49 °

Table 5.2: Nose geometrical parameters in millimeters

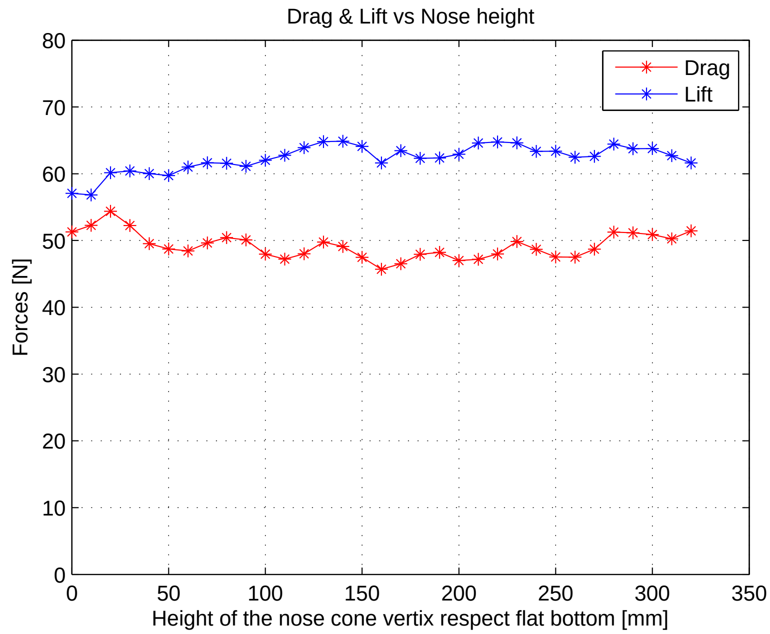


Figure 5.7: Nose height simulations

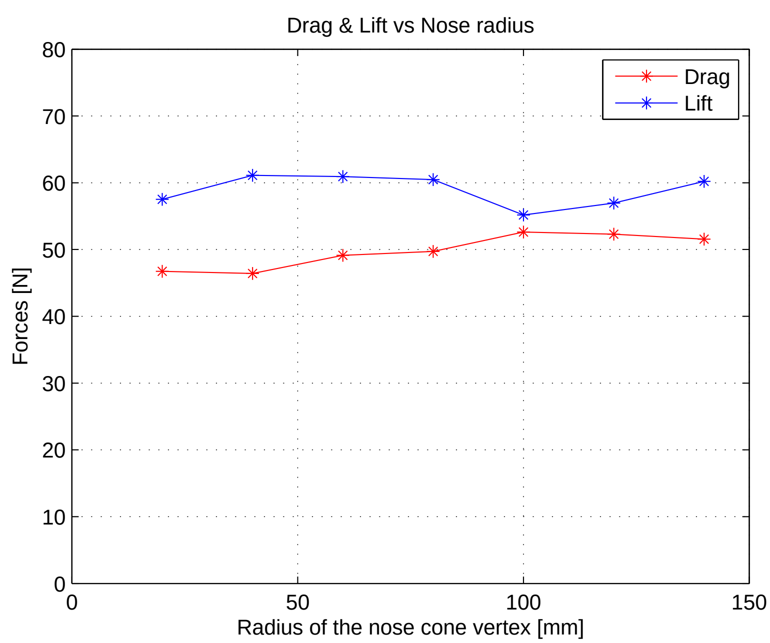


Figure 5.8: Nose radius simulations

The width of the nose is imposed by the chassis and the condition that in case the team want to include after this design a wing, a parallel wall respecting the XY plane, to attach that wing is needed. In figure 5.9 we can see the model A and in figure 5.10 the model B of the nose, that are the better models.

Model	Drag	Lift	Lateral
Nose A	51.287 N	57.058 N	0.155 N
Nose B	45.709 N	61.606 N	-0.171 N

Table 5.3: Nose cone design resume

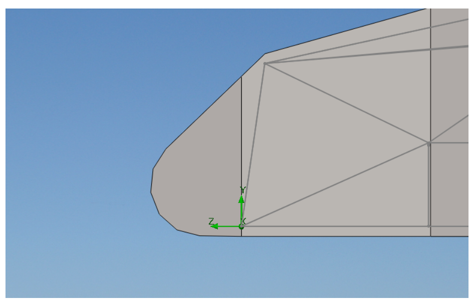


Figure 5.9: Nose model A

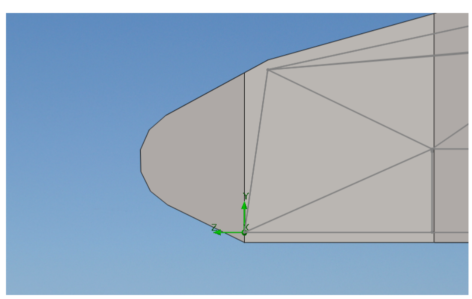


Figure 5.10: Nose model B

For the final design it was chosen the model A because in the reality, with the ground, the air flow goes less undisturbed flow to the rear part with this model. This means less drag force for the complete car.

As the rules said the nose cone has to have an element to attenuates the impacts in case of collision. This element can be design by the teams or just take the model that the organization gives. In this case, taking into account that is the first car designed and it has to be invested resources in other parts, the team made the decision of use the staff design without modifications.

For this reason, the division that most affected is the aerodynamic one. As can be seen in figure 5.11, the rules recommend that the impact attenuator has to cover almost all the square tubes and have a diagonal bar or a cross behind it. [Appendix B]

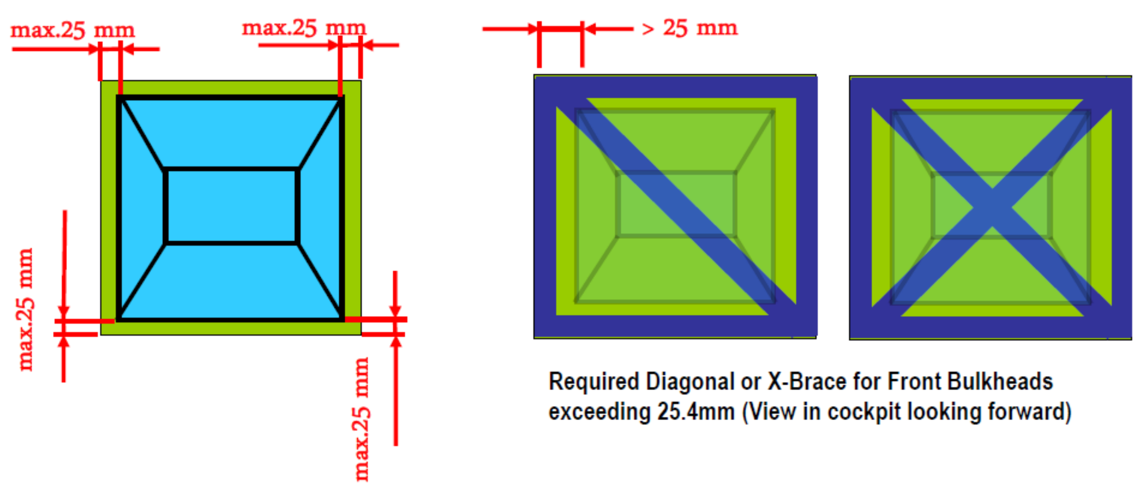


Figure 5.11: Attenuator diagram

5.7 Diffuser

The diffuser is the most important part of the car because the fluid that is going quickly below the bottom of the car it has to slow down and this situation generates drag and depending how smooth the diffuser is, it can generate lift or downforce.

In the figure 5.12 it can be seen the geometrical parameters that were considered to design the diffuser the horizontal length (DL) and the vertical height (DH) related both of them, with the angle respecting to the horizontal plane.

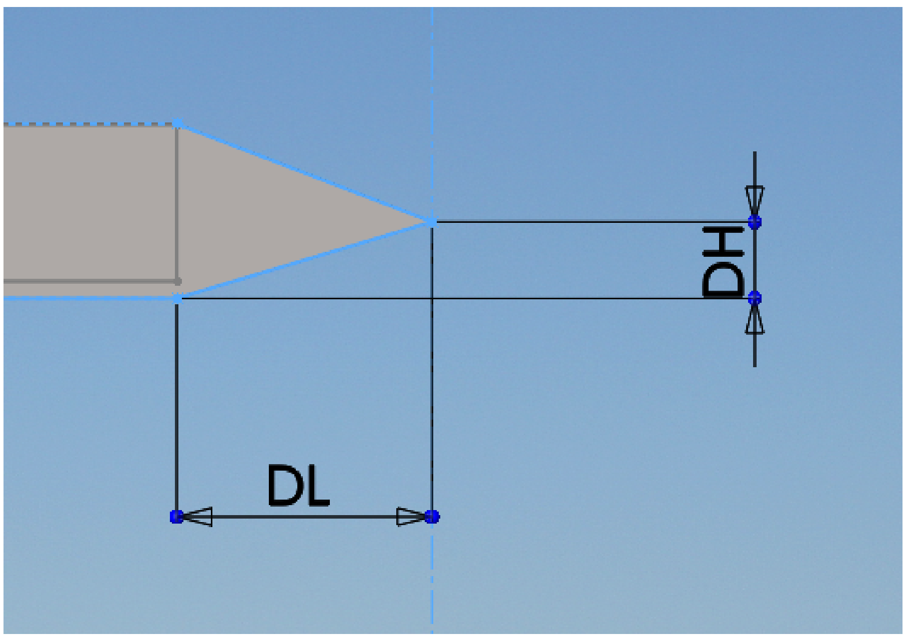


Figure 5.12: Diffuser geometrical parameters

Also, it was analyzed the use of small endplates in the diffuser but, they do not generate any benefit in the behavior or even getting worse. In figure 5.13 it is shown a diffuser curve plates that can slow down the air from the car bottom, but for this model, it worses. In case of figure 5.14 it neither works but they were a straight plates.

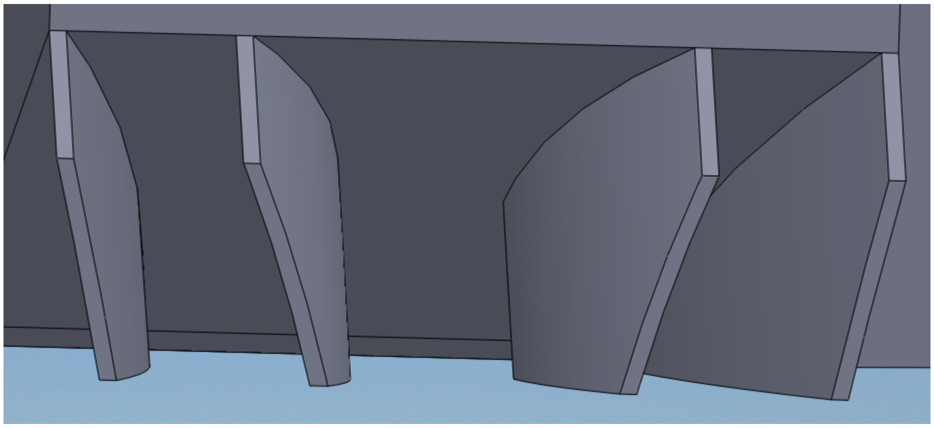


Figure 5.13: Diffuser curve plates

The plates are used to provide rigidity to the element and to reduce the creation of large vortices in the wake of the car.

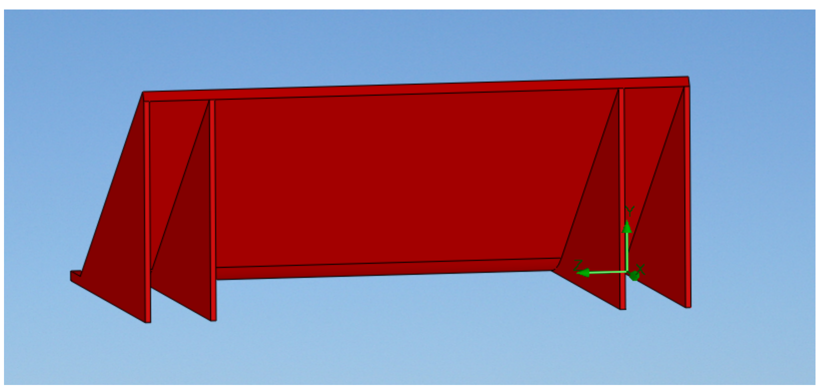


Figure 5.14: Diffuser straight plates

In figure 5.15 it can be seen, that it was modeled a bottom with channels to generate vortices in the turbulent flow and slow down in the rear part the flow. This vortices changes the flow into turbulent but they avoid the detachment of the laminar flow. However, this also worse for the car behavior.

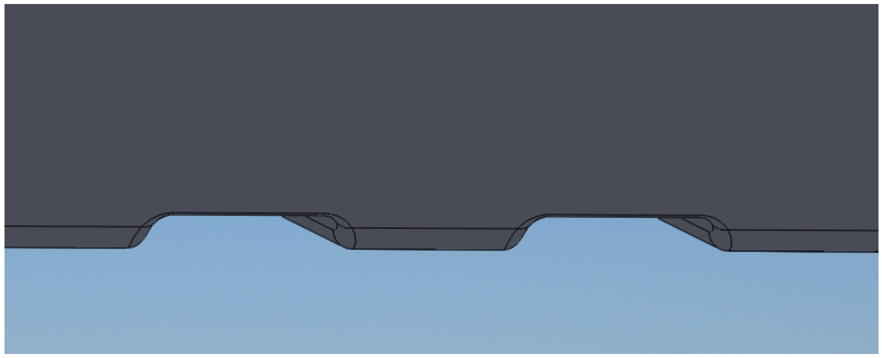


Figure 5.15: Non flat bottom shape

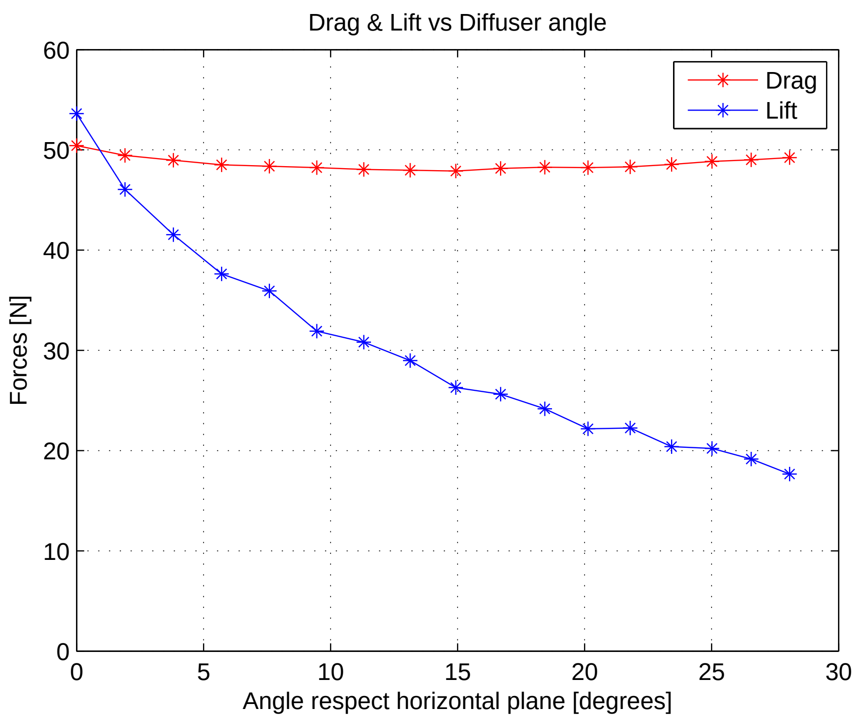


Figure 5.16: Diffuser height simulations

In figure 5.16, the variation of the drag force is constant but the lift is reducing as much the angle is bigger, this is beneficial to get finally a downforce. In the table 5.4 it is describe the dimensions of the diffuser.

Model	Angle	Drag	Lift	Lateral
Diffuser	28 °	49.223 N	17.667 N	-0.028 N

Table 5.4: Diffuser cone design resume

5.8 Air intakes

This element influence in the car behavior because it increase the frontal area of the car, meaning, the increment of drag force. Also, because it is a way to release heat from the engine changes the conditions of the flow that go throw it.

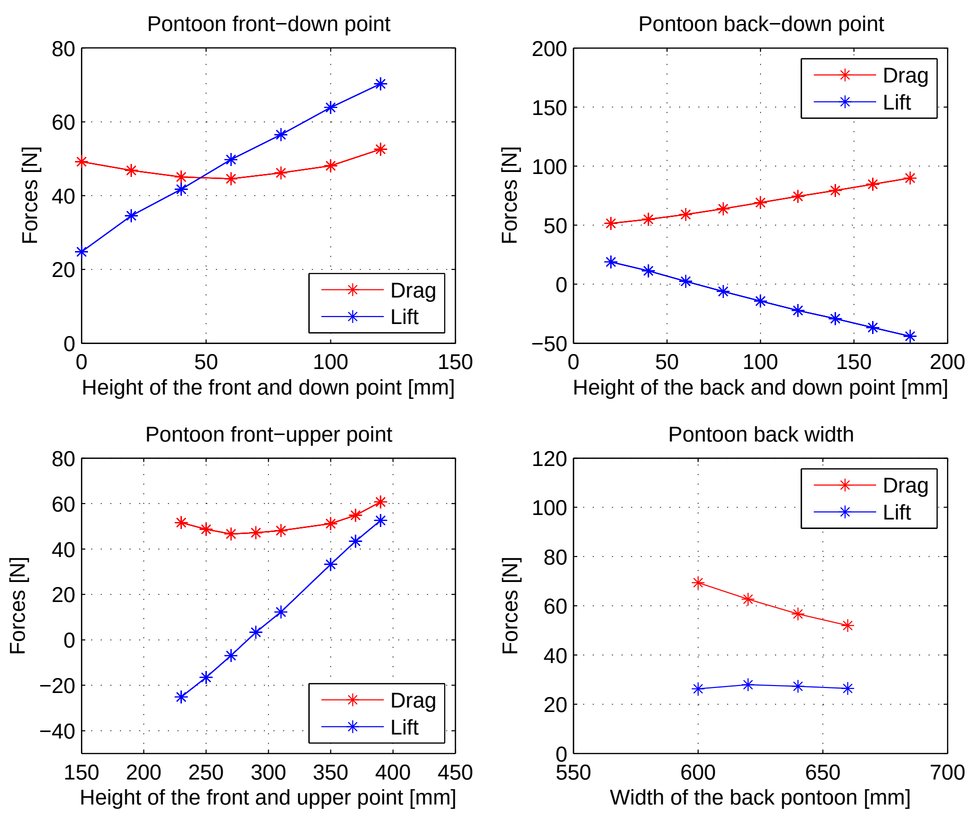


Figure 5.17: Pontoon dimensions analysis

The last graph of figure 5.17 shows that is better to open the las part of the pontoon to slow down the air, however this air flow hits with the wheel if it is considered it and then instead of improve the behavior, get worse.

Model	Front height	Back height	Slope	Front width	Back width
Pontoon A	270	290	5.1 °	386	293
Pontoon B	270	250	8.5 °	386	293

Table 5.5: Pontoon geometrical parameters in millimeters

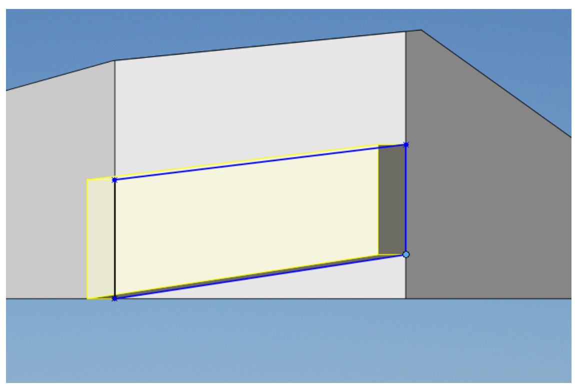


Figure 5.18: Pontoon A

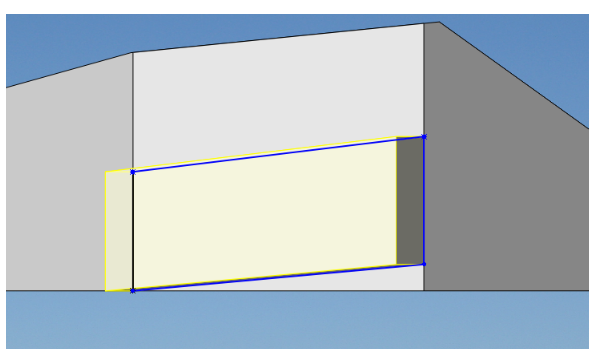


Figure 5.19: Pontoon B

The figures 5.18 and 5.19 show the differences in the air side intakes. This variations only changes when the angle of this pontoon goes up. This is enough to vary the forces significantly. In table 5.5 are specified the dimensions of the pontoon and in table 5.6 the resultant forces of the models.

Model	Drag	Lift	Lateral
Pontoon A	52.497 N	-26.109 N	0.192 N
Pontoon B	60.358 N	-41.953 N	-0.033 N

Table 5.6: Pontoon design resume

As figure 5.17 shows, it was chosen a front down point at the same height that the bottom, and the upper one also as low as possible but keeping enough area to cool down the radiators. In case of the back point is better to increase it height until a point were the drag goes up quickly. All the point can be seen in 5.20.

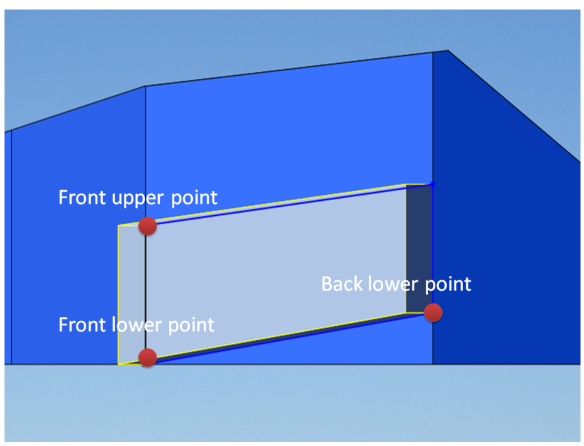


Figure 5.20: Pontoon geometrical parameters

For the final design it was selected the model A because despite of the increment in drag in a few newtons, it almost double the downforce that gives. Then for the final car it will be have more grip.

It is assumed that the engine cooling system consists of two radiators are placed one on each pontoon based on the calculations of the engine division. These radiators, as can be seen in figure 5.21 are placed at 45 degrees to make the frontal area smaller and they do not lose too much efficiency to transmit the heat to the air.

A motorbike usually is cooled by air, however a Formula Student car using a motorbike engine, can not be cooled by air because the engine position in the car that do not have clean air and enough speed. The teams put radiators in the most down part that can receive cool air, that are sidepods. Although the speed are low, it is not advisable to have a fan to move the air faster because this fan can not generate any downforce, in case of doing it, the team is disqualified.

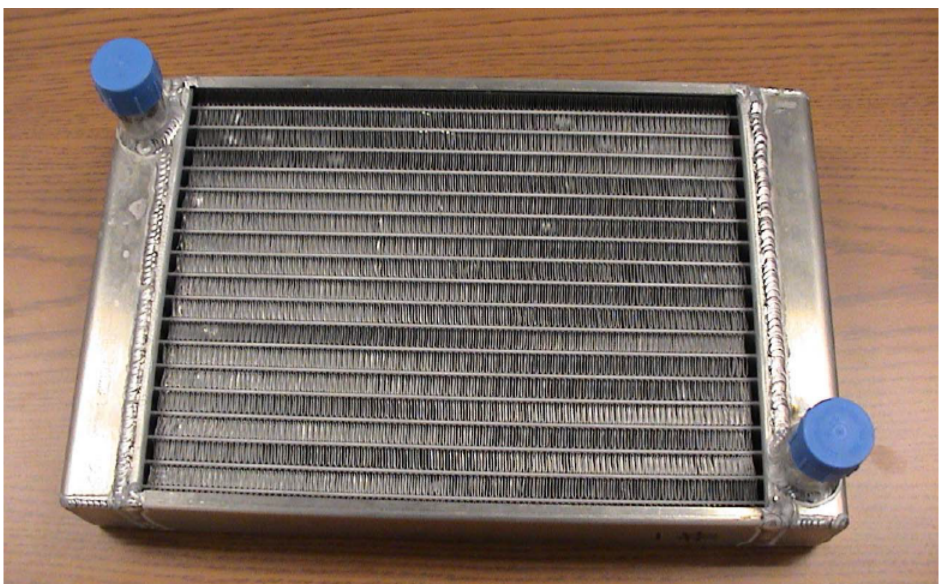


Figure 5.21: FSAE radiator

It has also been considered placing the radiators in a more rear position than the current to move the center of gravity of the car a bit more to the rear axle. This idea has been discarded because it may cause interference with rear suspension which is not yet designed. Also, it has to be balanced the weight in each axle and as the engine and the driver are placed at the rear axle, the radiators has to be placed in a front position.

5.9 Final design

The previous analysis have been done with sharp edges because it is easier to compare models without radii varibles that influence. But for the real car is much better for improving performance and for safety reasons to round the shape. The results that have been done are describe in table 5.7.

It has been apply a round operation of radius 10 mm in all the edges that allow that. The decrement of the drag is due to the reduction of the total frontal area and this do not affect to lift forces. The lateral force also decrease due to the less instability because of less vortices produce by the edges.

Model	Drag	Lift	Lateral
Sharp	65.309 N	-96.055 N	0.200 N
Rounded	63.683 N	-96.269 N	0.0468 N

Table 5.7: Sharp vs rounded designs resume

It was done a comparison between the mesh quality and the computing time as can be seen in table 5.8. The reason to analyze the geometrical parameters with a medium quantity of cells is due to time. All the calculations were done with a four core processor and 2.8 GHz using all its capacity. In case of doing all the simulations along 11 hours changing just a parameter each time, this thesis could not be possible to do in the supposed time.

Also for the competition, that the differences between cars are measured in few seconds and the top speeds are low comparing to Formula 1 cars. The results obtained by low quantity of cells are also ignore due to in the boundary layer the cells are much bigger than the layer. This produces some errors that have to be avoided.

For all of this, it was chosen for non critical part like the nose cone an approximately 400,000 cells resolution and in case for delicated parts in terms of the boundary layer such as the diffuser it was done with approximately of 1,000,000 cells.

Cells	Time of computing	Drag	Lift	Lateral
78,058	00:07:11	60.279 N	-86.335 N	0.270 N
173,654	00:22:15	64.899 N	-96.174 N	0.202 N
369,380	00:44:15	61.806 N	-93.098 N	0.457 N
484,286	01:04:03	60.324 N	-101.956 N	0.290 N
980,083	02:25:14	59.508 N	-94.787 N	0.261 N
2,688,447	11:30:08	59.887 N	-97.857 N	0.150 N

Table 5.8: Mesh results comparative at 90 km/h

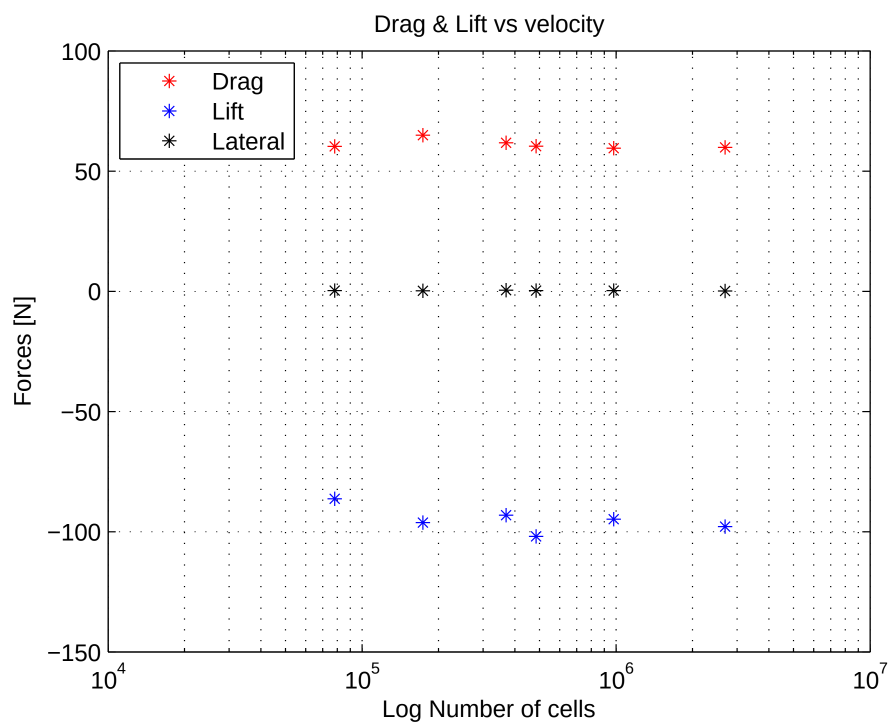


Figure 5.22: Comparative results between number of cells at 90 km/h

In figure 5.23 it is showed the final mesh, that is fine close to the chassis and coarse far respecting the bodywork. It was done automatically by the software.

Also, it is useful for the team, how much forces experimented the car when it goes at different speeds, because 90 km/h is a control value to compare the forces. The comparison between forces at different speed is shown in table 5.9.

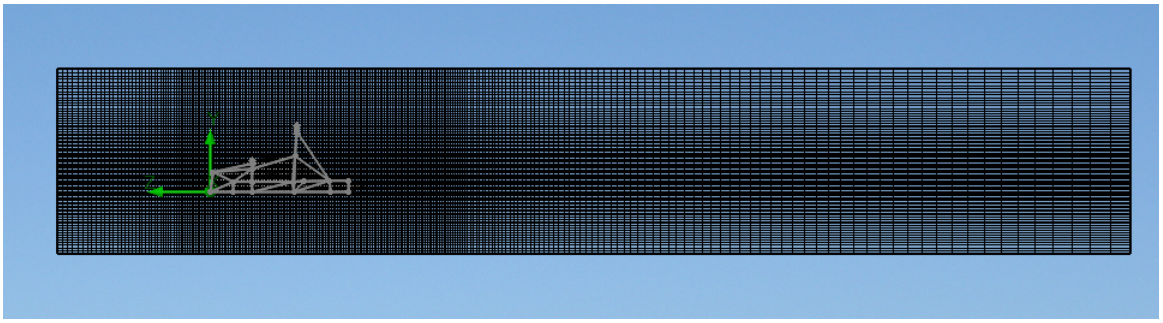


Figure 5.23: Final mesh

Speed [km/h]	Drag	Lift
20	3.201	-5.046
40	12.851	-19.327
60	28.884	-42.826
80	51.369	-76.382
100	80.297	-118.953
120	115.628	-171.741
140	157.292	-233.708

Table 5.9: Forces comparative vs speed

In case of the figure 5.24 is the translation of the table 5.9 to a graph where it can be seen perfectly that the forces varies following a parabola curve. In table 5.10 it can be seen the drag and lift coefficients average values from the forces versus speed.

C_d	0.303
C_l	-0.455

Table 5.10: Drag and lift coefficients

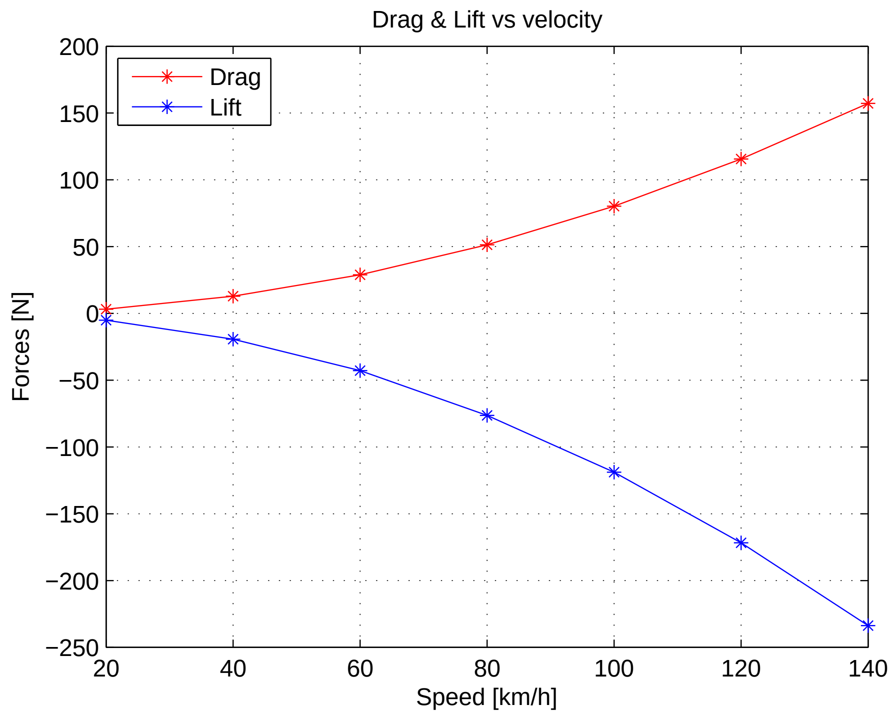


Figure 5.24: Forces versus speed

The figure 5.25 is the representation of the pressure distribution of the air along the bodywork. It is maximum in the front parts because the air stops there and minimum where the air has to go fast.

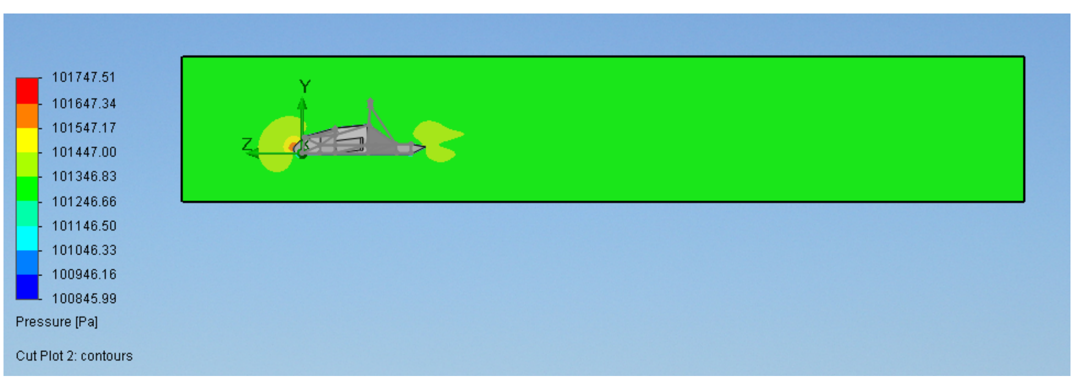


Figure 5.25: Final design pressure distribution

In figure 5.26 it can be seen a velocity distribution of air which is zero at the nose cone and maximum below and above the bodywork and from there, it slows down until far from the car.

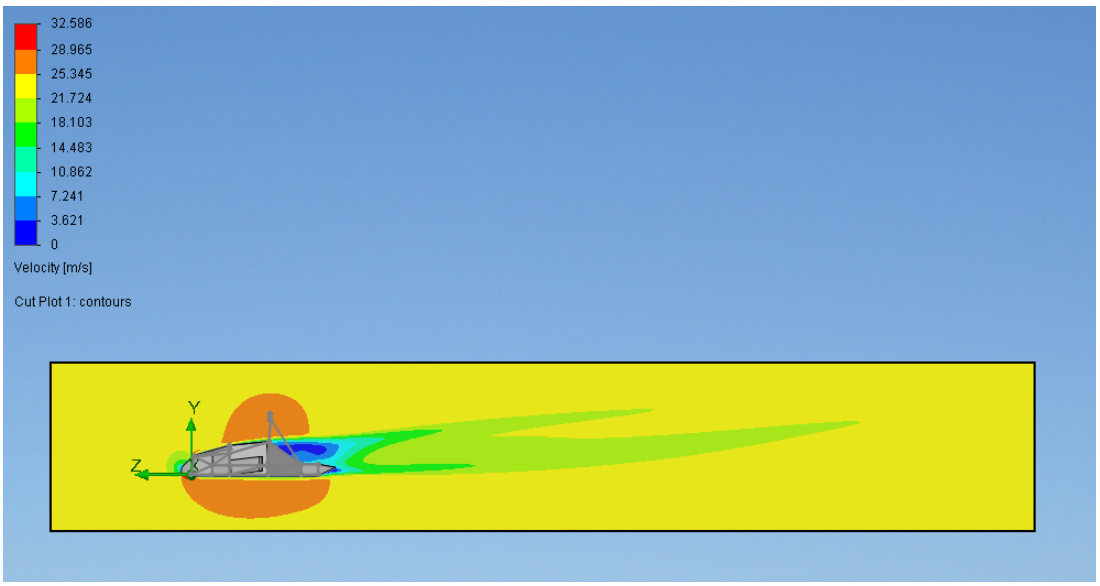


Figure 5.26: Final design velocity distribution

Finally, the figure 5.27 shows the final bodywork with university official colour and having the ground below it.

As it has been develop previously the final bodywork will have the nose model A, the diffuser with maximum angle and the pontoon model A. All the bodywork has been designed trying to get closer as much as possible to the tubes for minimizing the frontal area and having smooth changes between surfaces to reduce drag.

External elements such as wheels, suspension or wings have not been taking into account because they are in the developing phase and non definitive.

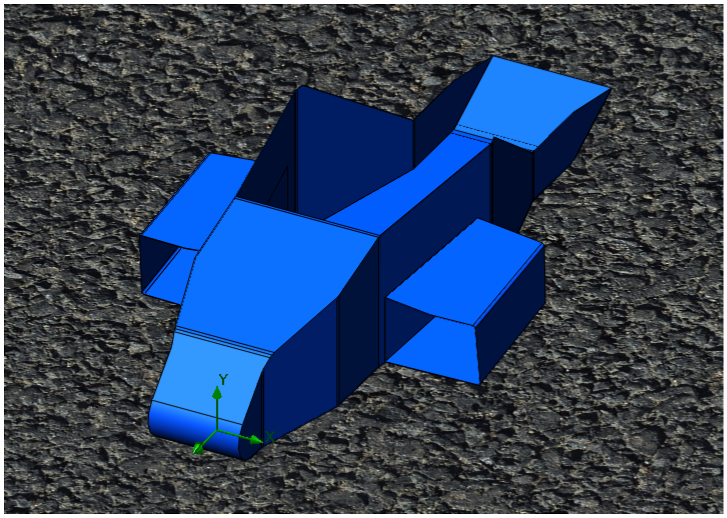


Figure 5.27: Final design

5.10 Chassis supports

The bodywork has to be attached to the chassis to avoid that the car at high speed looses it. There are many ways to do it, some are temporal fasteners and others are permanents.

For this car, that is going to be tested many times, because it is the first one that the team and the university makes, and assuming that it will appear some breakdowns and failures as all prototypes have. It has been selected a temporal fastener.

This joint consist in a M8 screw with a M8 nut. The screw has a flat head to avoid protruding above the bodywork. And the nut will be soldered to the tubes or to plates that they will be soldered to the tubes as it is shown in figure 5.28.

The exactly position where the nuts and screws are placed is not completely defined because the chassis at the time where this bachelor thesis is done it has not been finished. The external measurements are definitive but how the tubes are place and cross themselves will be change.

Figure 5.28: Nut soldered to a plate

5.11 Manufacturing process

To select materials was taken into account its price, easy of construction and weight in order to make the car faster with a reasonable cost for the team.

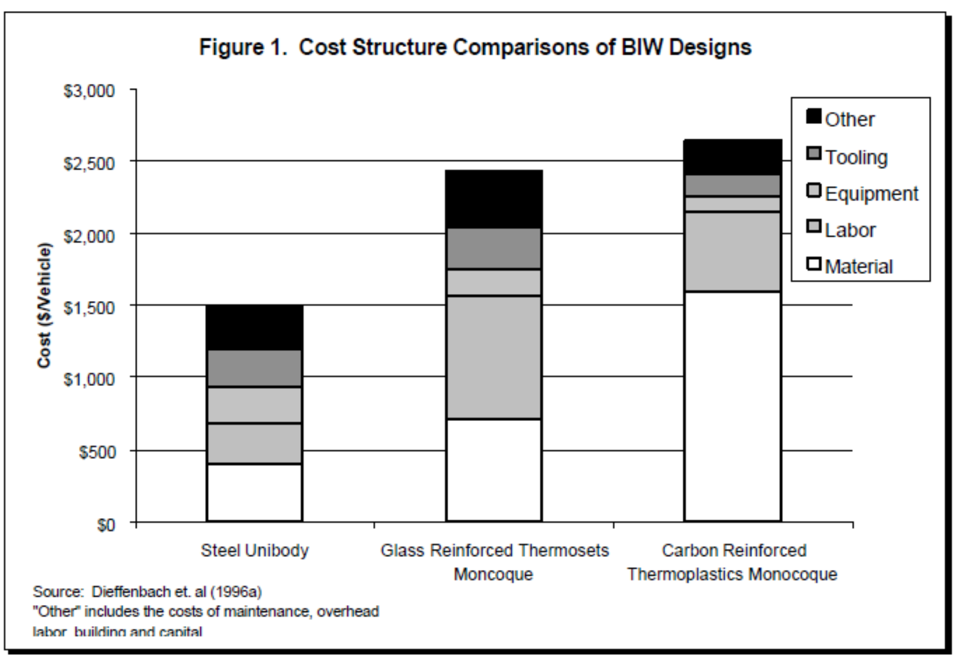


Figure 5.29: Composite comparation versus cost

In figure 5.29 it is shown a comparative between the cost of manufacturing bodyworks if they are made from steel, glass fiber or carbon fiber. In case of steel, it is the cheapest but doing a bodywork of steel for a car that the total weight excluding the bodywork is 320 kg, it can be almost the 50% of the weight.

In case of carbon fiber, it is the most expensive and it is need an oven to cure it, so it is more difficult to build the bodywork with carbon fiber. Finally the suitable material for the team is the glass fiber.

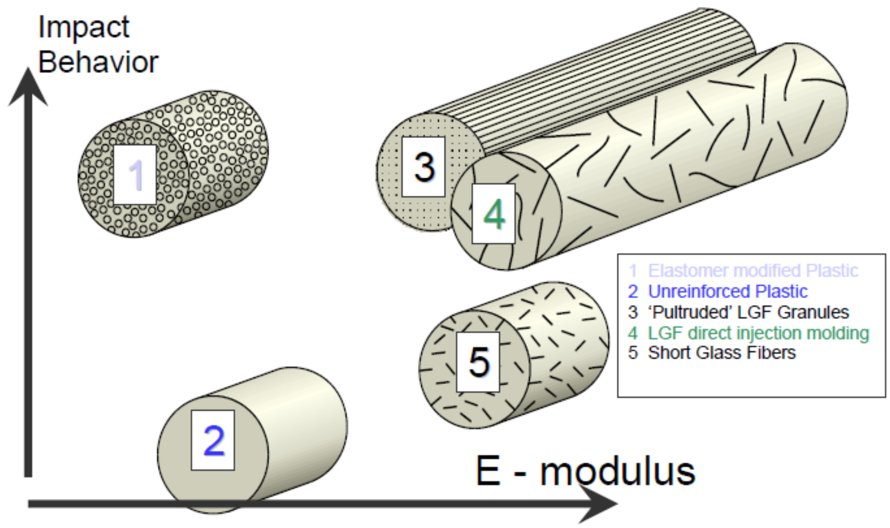


Figure 5.30: Fibers types

In case of build a piece with composite materials, it is important to choose how it works depending the fibers orientation and the proportion of the fibers and the matrix. In figure 5.30 it is shown a comparative.

The construction of the models is not easy, it has been followed some steps to do the process correctly. For manufacturing it is needed some materials:

- Lab coat.
- Gloves.

- Safety glasses.
- Wood stock.
- Wooden planks.
- Screws.
- Drill.
- Bright wood sealer.

Steps for constructing the pieces:

- Put on the lab coat, gloves and safety glasses to protect clothes and skin from chemicals during the turning process.
- Build a female mold using wood planks, screws and a drill for the body. It must be a female mold for the outside of the car because it is smooth. The mold is larger than the final product. The mold should be about the width of a wooden board over across the way it was planned for the final product.
- Give a hand of polished wood sealer to the mold. This will create a smooth layer to which the fiber and resin do not stick.
- Apply the glass fiber and the resins to make the piece and leave it until it dries.

Figure 5.31: Glass fiber sheets

In the market it can been selected glass fibers that they can be disposed without any order or buy a long sheet that have interwoven fibers. In figure 5.31 it is shown an intermediate form, that consists in small sheet that can be stick crossing the fibers for making it stronger.

When the mold is builded, it is covered by glass fiber sheets and the epoxy matrix. It has to be waited until it dries and in figure 5.32 it can be seen this process. After it is completely dry, the molds are removed and it is polished the surface for doing it smoother.

Figure 5.32: Glass fiber mold

5.12 Testing

Once it is manufactured the prototype it will be done a series of tests to analyze. In case of for the competition it has been done by the teams as their money and their facilities allow it.

The teams usually test the engine, brakes and suspension. However, teams that consider the aerodynamics important, and also, have money for constructing them for their cars. They develop wings and endplates that has to be tested in the track to analyze their efficiency and behavior.

Chapter 6

Structural analysis by Finite Elements

6.1 The Finite Element Method

The Finite Element Method is a numerical algorithm to find an approximation to differential equations solutions.

The FEM was develop in 1943 by Richard Courant, who used the Ritz method of numerical analysis and minimization to obtain approximate solutions to a system of vibration. After, a paper published in 1956 established a definition of the numerical analysis. The paper focused on the rigidity and deformation of complex structures. [7]

With the first computers introduce matrix calculus of structures. This part of the discretization of the structure in bar-line elements of known stiffness versus displacement of its nodes. This system of equations is outlined as follows:

$$f = Kd \tag{6.1}$$

Where:
f is a vector of the forces.
K is the rigidity matrix.
u is usually the unknown variable, the displacements.

The FEM is used with computers, than can solve easily this equations in very few time even the geometry is complex, the time depends mainly in how fine or coarse is the mesh. The computer will solve the constitutive equations obtaining in this way the displacements and after that stress and strains.

This method obtains the solution from a continuum media diving it in small portions that are connected each other with nodes called the mesh. The computer solves the equations for each node and applies compatibility conditions to near nodes. Then, the number of equations of the system is proportional to the number of nodes, more nodes, more equations, more time. [8]

An important property of the method is convergence, if partitions are considered finite elements successively thinner, the calculation numerical solution converges rapidly to the solution of the equations.

Before computers, the calculus of structures with iterative method implies some weeks to do, because of using many equations. This imply a large cost because spend time instead of optimizing the structure the engineers have to solve it.

Nowadays, with computers, this is much easier and fast, the post-processor print in the screen the results with colors. This results have high quality.

6.2 Algorithm

The MEF algorithm is:

The independent variable domain must be divided by a partition into subdomains, called finite elements. Associated with the above the partition builds a finite dimensional vector space called finite element space. Being the approximate numerical solution obtained by a linear combination of finite element in that vector space. [9]

It is obtained the projection of the original variational problem on the finite element space obtained from the partition. This results in a system with a finite number of equations, although generally with a high number of unknowns equations.

The number of unknowns is equal to the dimension of the vector space of finite elements obtained and, in general, the greater this dimension the better the obtained numerical approximation. The last step is the numerical solution of the system of equations.

The above steps allow building a calculus problem in a linear algebra problem. This problem generally arises on a vector space of dimension not-finite, but that can be solved about finding a projection onto a finite dimensional subspace, and therefore with a finite number of equations (although in general the number of equations will be high typically thousands or even hundreds of thousands).

The finite element discretization helps to build a simple projection algorithm, also achieving the solution by the finite element method is generally accurate in a finite set of points. These points usually coincide with the vertices of the finite elements. For specific resolution of huge system of general algebraic equations can be used conventional methods of linear algebra in finite dimensional spaces. [10]

6.3 Calculation

The material selected is glass fiber that usually is used in the industry to do some car pieces of the bodywork. Its characteristics are resumed in table 6.1.

The loads and the boundary conditions are applied using the criteria that is showed in figure 6.1.

The conditions for applying forces are the following:

- The drag and lift forces are multiplied by a security factor of 1.5.
- All faces that have contact with a frontal flow has applied a uniform distributed drag force.
- All faces that have an horizontal flow have applied a uniform distributed lift force.
- The points to attach the piece have been choose according to a probable point where the joints can be placed.

Property	Value
Fiber volume fraction	0.43
Density (g/cm3)	1.85
Longitudinal modulus (GPa)	26
Transverse modulus (GPa)	22
Shear modulus (GPa)	7.2
Poissons ratio	0.13
Longitudinal tensile strength (MPa)	400
Longitudinal compressive strength (MPa)	350
Transverse tensile strength (MPa)	380
Transverse compressive strength (MPa)	280
In-plane shear strength (MPa)	45

Table 6.1: Glass fiber properties

A conservative thickness of 2 millimeters has been chosen. It can be reduced more than a half in many points during the manufacturing. The software do not allow to do it.

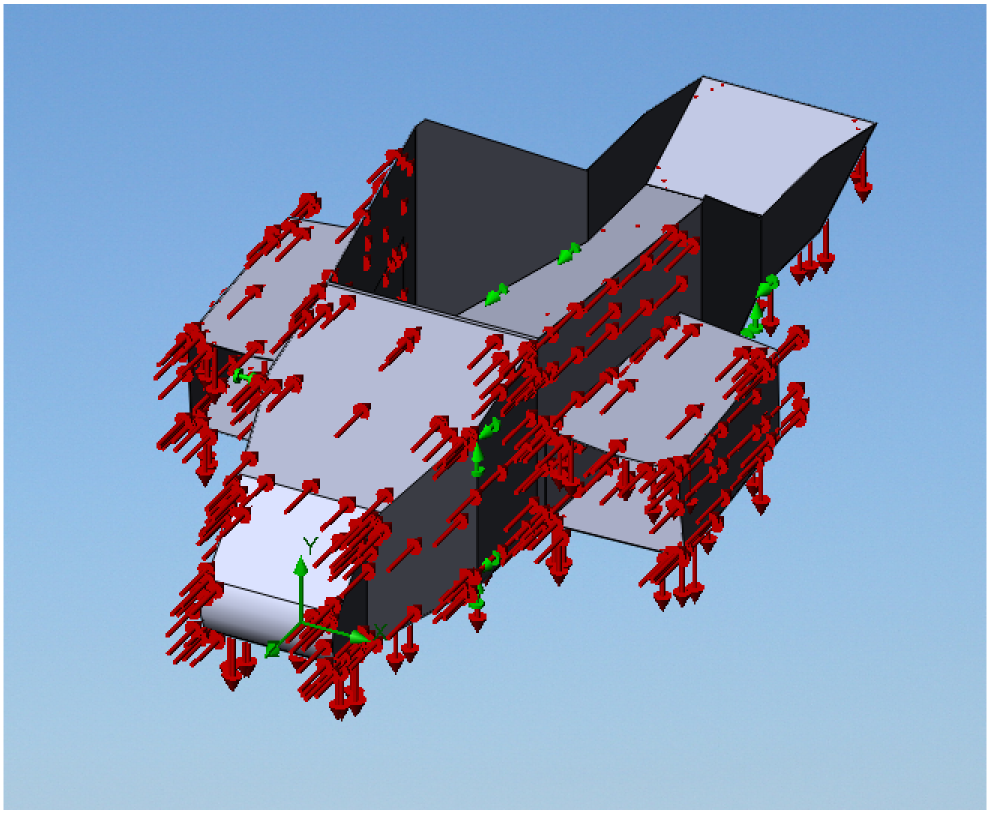


Figure 6.1: Boundary conditions applied

SolidWorks creates a square shape mesh automatically when it going to solve the system. It was considered to do with the finest mesh that the program allow, showed in figure 6.2 , increasing the computing time but obtaining better results.

In figure 6.3 the maximum tension that the piece have is 13.7 MPa much lower that the limit that appears in table 6.1. This means, the piece is completely in the elastic region, so its behavior it is predictable.

The critical points are close to the boundary conditions and contact points between the body and the pontoons. The first problem can be solved using a joint with a big contact area and the second one can be minimize during the manufacturing process smoothing the transition between the body and the pontoons.

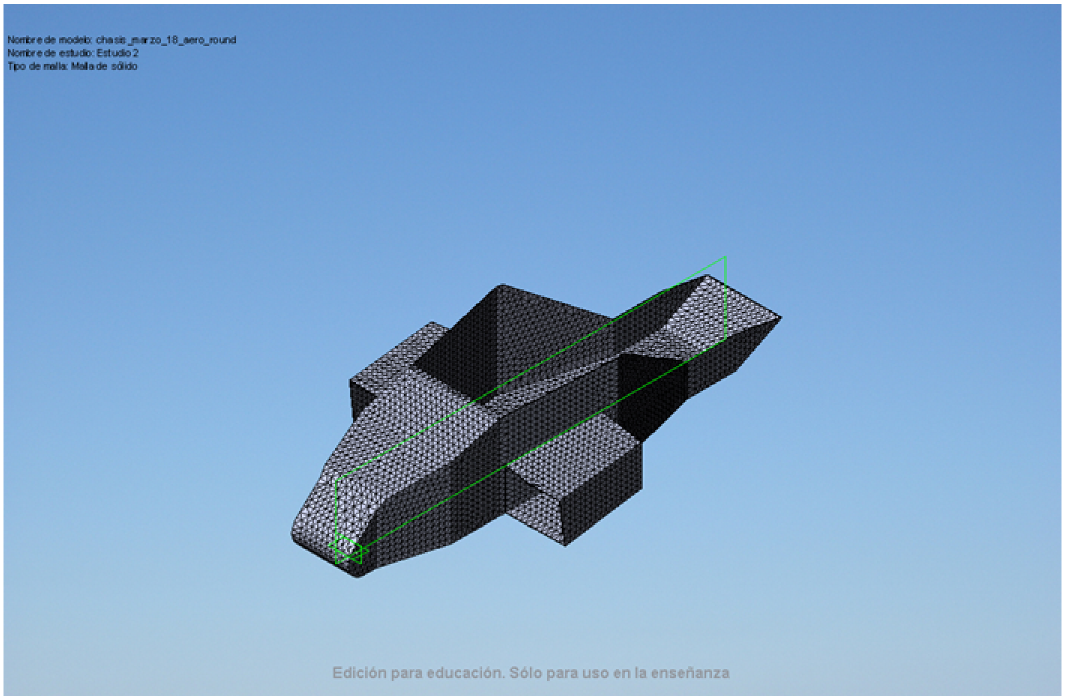


Figure 6.2: Mesh discretization

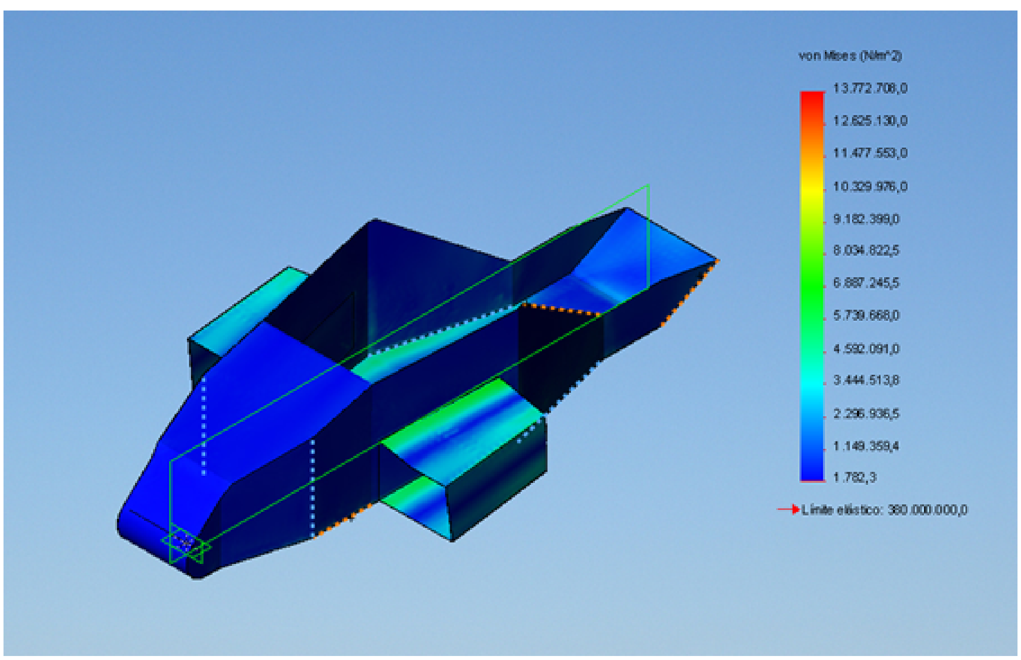


Figure 6.3: Tension distribution

In case of piece deformation, and it is measured by the displacements as can be seen in figure 6.4, the pontoons have the maximum displacement because it has a great area where the drag and the lift are applied. This problem can be solve increasing the strength of the composite in that regions.

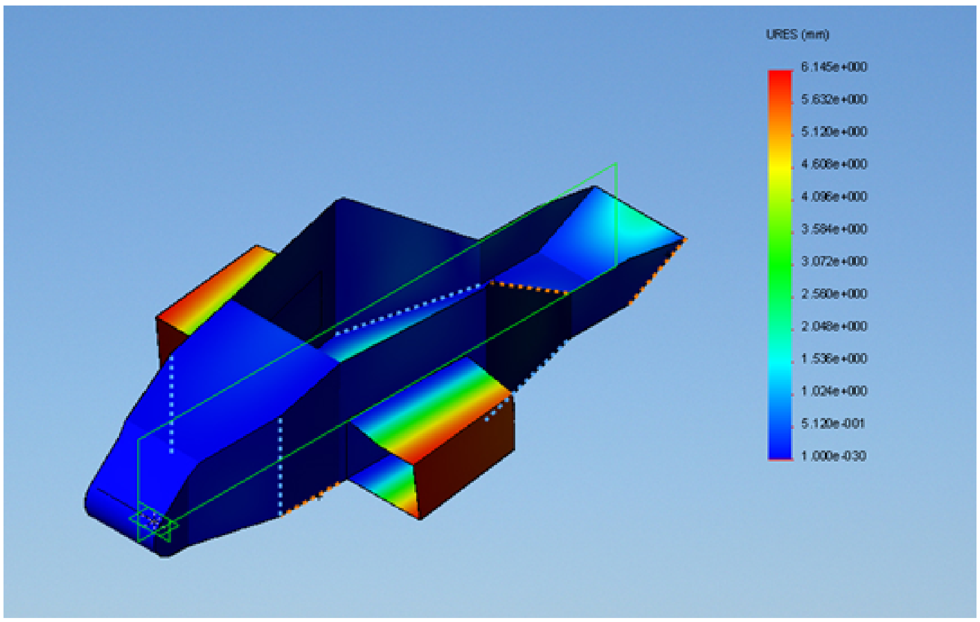


Figure 6.4: Displacement distribution

Chapter 7

Cost estimation

7.1 Construction at the university

This is the cost estimation if the team choose to construct in the university:

Element	**Cost unit**	**Units**	**Total**
Glass fiber filaments	6.92 €/ kg	4 kg	27.68 €
Epoxy resine	29.28 €/ kg	12 kg	351.56 €
Wood			150 €
Tools			400 €
Total materials and tools		20 kg	829.04 €
Design	35 €/ hour	41 hours	1435 €
Total engineering cost		41 hours	1435 €
Making mold	20 €/ hour	6 hours	120 €
Applying glass fiber and resin	20 €/ hour	4 hours	80 €
Cure in oven	20 €/ hour	2 hours	40 €
Total making cost		12 hours	240 €
Final cost			**2604.04 €**

7.2 External manufacturing

In case of the team choose to externalize the manufacturing process there are some firms that make glass fiber pieces for automotive industry:

- ESCOPrem. More info: (*http://www.escoprem.com/fibra-de-vidrio.asp*)
- MOLDEO Y DISEÑO S.L. More info: (*http://www.moldeo.com/home.htm*)
- BIHER Plásticos. More info: (*http://www.biherplasticos.com/es/index.asp*)
- REFIGAL S.L. More info: (*http://www.refigal.com/*)
- Fividrio. More info: (*http://www.fividrio.es/home.html*)

They have not been contact for getting a budget with the companies yet because the marketing division of the team is designing a strategic for sponsoring, and they do not want interferences before they finish.

Chapter 8

Schedule

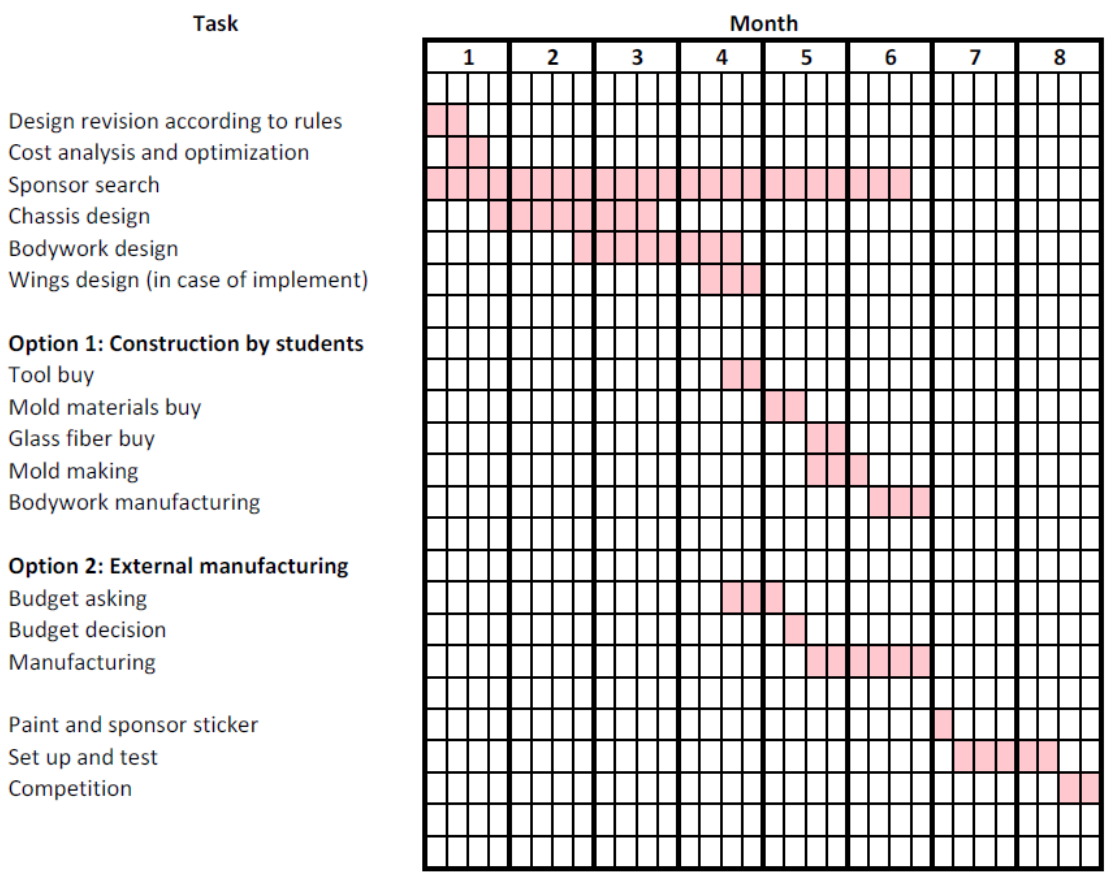


Figure 8.1: Team planning

Chapter 9

Environmental impact

Since some time it has been talked about the climate change and global warming. And in case of the transport industry, it has been one of the most discussed, because its contribution to CO_2 and NO_x are a great quantity of the total.

Some international laws or agreements have been developed trying to stop this emissions. These, consist in the prohibition of some components like lead or taxes for cars that pollute more.

For reducing this emissions is not only important improving the engines but build lighter cars, with less weight, using instead of metal sheets or wood for bodywork, it is increasingly the use of composite materials as glass fiber or carbon fiber which give more strenght with less weight. This materials at the beginning were very expensive and nowadays have less prices but still too high for the common people.

The recycling of this composites is separated from the rest of the other car materials because some of them can be use again using chemical technics with nitric acid, or thermal techniques in a controlled environment, can be applied to these composites with epoxy base.

The weight reduction not only decreases the fuel consumption at the same speed, in case of competitions car, it helps for running fast with the same engine. If the construction of a formula car bodywork had been full of carbon fiber comparing with wood or aluminium chassis, and considering that these car weight approximately 300 kg, it is 20% approximately savings.

Chapter 10

Conclusions

This bachelor thesis aim is making a design that the university Formula Student team can used in a nearly future in the competition. Because of this, the goals of doing a simpler design have been completed, not only in the manufacturing process, also in the assembly process that the team have to repeat several times for reparing operations.

The bodywork is not an essential part of the car, because the car can move without it. However its benefits improve that car behavior doing it faster and with more grip. Also it gives to the car protection to the steering and the driver in case of raining.

The use of the forces graph respect to the speed it can be a useful tool to the team to analyze how much power the car needs in each competition event and doing some modifications to have a suitable aerodynamic car when it will be build.

And the variation of the forces depending some geometrical parameters helps to team to choose and change parts if there are modifications in chassis or the whole car.

The specific objectives of this project have been met:

- Nose cone: it has been parameterized the geometrical parameters. It has been modeled designs with variations and analyzed how they work. Finally it has been reach a final design optimized.
- Side pontoons: it has been develop a parametrical side pontoons that can be implemented when the engine division calculate the cooling system. Also after some model analysis has been acquired an optimized design.
- Diffuser: it has been reach a simpler design using two parameters that they were simulate and optimize the results.
- Complete bodywork: it has been design trying to be as close as possible to the tubes for reducing the maximum of the frontal area and the material to manufacture.

Although it has been design for using glass fiber as material and doing its structural and fluid analysis, it can be proposed a cheaper bodywork in case of the team do not have enough money for it. This bodywork consists in covering the tubes with plastic film because the bodywork loads are not great and it is much cheaper than glass fiber.

Chapter 11

Future development lines

This bachelor thesis is focused in find a solution to build a bodywork with a good behavior in the competition, cheap to the team and easy to work with.

For next designs it can be improve:

- More aerodynamic shape: knowing how the fluid moves in this design, it can be done another design using complex curvatures and reference points to get a more efficient bodycar.
- Lighter materials: in this case, it was considered glass-fiber because is relative cheap and easy to manufacture but the best option is carbon-fiber, although it is very expensive.
- Include wings and another elements: there is another bachelor thesis working on wings for the team, then, it could be a good choice to include in the bodycar to analyze together.
- Analysis with more boundary conditions: it can be included, when the team defined their position, shape, and another parameters, for instance, the wheels, the engine or the radiator.
- Compare CFD with wind tunnel: the university have a wind tunnel, it can be improved the design comparing the computer solution with a model in the wind tunnel.
- Build the car and run with it to analyze the real world all the elements and parts.

Bibliography

[1] William F. Milliken and Douglas L. Milliken,
Race Car Vehicle Dynamics, First Edition, SAE International,1995. ISBN: 1-56091-526-9

[2] Joseph Katz, *Race Car Aerodynamics - Designing for speed*, First Edition, Benthey Publishers,1995. ISBN: 0-8376-0142-8

[3] Frank M. White, *Fluid Mechanics*, Sixth Edition, Mc Graw Hill,2008. ISBN: 978-84-481-6603-8

[4] Tu, Yeoh, Liu, *Computational Fluid Dynamics* Butterworth- Heinemann , 2008. ISBN: 978-07-506-8563-4

[5] Hirsch, *Numerical Computation of Internal and External Flows* Butterworth-Heinemann , 2007. ISBN: 978-07-506-6594-0

[6] Jose Manuel Gordillo, *Introduccion a la aerodinamica potencial*, Edicion Primera, Paraninfo. ISBN: 978-84-9732-994-1

[7] Bulletin of American Mathematical Society, *Variational methods for the solution of problems of equilibrium and vibrations*, 49, 1-43. 1943.

[8] CIMNE, *El método de los Elementos Finitos*, Mc Graw-Hill Barcelona .1994.

[9] John Wiley, *Stress Analysis* London. 1966.

[10] Mc Graw-Hill, *The Finite Element Method in Structural and Continuum Mechanics* London. 1967.

[11] Formula SAE
`http://students.sae.org/competitions/formulaseries/`
Last visit: 05/29/2013

[12] Society of Automotive Engineers
http://www.sae.org
Last visit: 05/29/2013

[13] Institution of Mechanical Engineers
http://www.imeche.org
Last visit: 05/29/2013

[14] Sociedad de Técnicos de Automoción
http://www.stauto.org
Last visit: 05/29/2013

[15] Escudería UC3M
https://www.facebook.com/escuderiauc3m
Last visit: 05/29/2013

[16] European Car Web
http://www.europeancarweb.com/
Last visit: 05/29/2013

[17] Classic Driver http://www.classicdriver.com/uk/magazine/
Last visit: 05/29/2013

[18] F1 Grand Prix
http://f1-grandprix.com/
Last visit: 05/29/2013

[19] European Car Magazine. Cheating Wind - Aerodynamic Tech and Buyers Guide: The Art Of Aerodynamics And The Automobile. Price, Ryan Lee. http://www.europeancarweb.com
Last visit: 05/29/2013

[20] F1 Technical
http://www.f1technical.net/
Last visit: 05/29/2013

[21] Formula 1 Dictionary
http://www.formula1-dictionary.net/nose_cone.html
Last visit: 05/29/2013

[22] Fuel Wasters
http://www.fuelwasters.com
Last visit: 05/29/2013

[23] Formula 1 Aerodynamics. Smith, Rich.
http://www.symscape.com/blog/f1_aero
Last visit: 05/29/2013

[24] Rapid Racer
http://www.rapid-racer.com
Last visit: 05/29/2013

[25] F1 - History of Aerodynamics: Evolution of aerodynamics.
http://www.f1network.net/main/s107/st22394.htm
Last visit: 05/29/2013

[26] The Drag Coefficient.
http://www.grc.nasa.gov/WWW/K-12/airplane/dragco.html
Last visit: 05/29/2013

[27] Beginner's Guide to Aerodynamics.
http://www.grc.nasa.gov/WWW/K-12/airplane/bga.html
Last visit: 05/29/2013

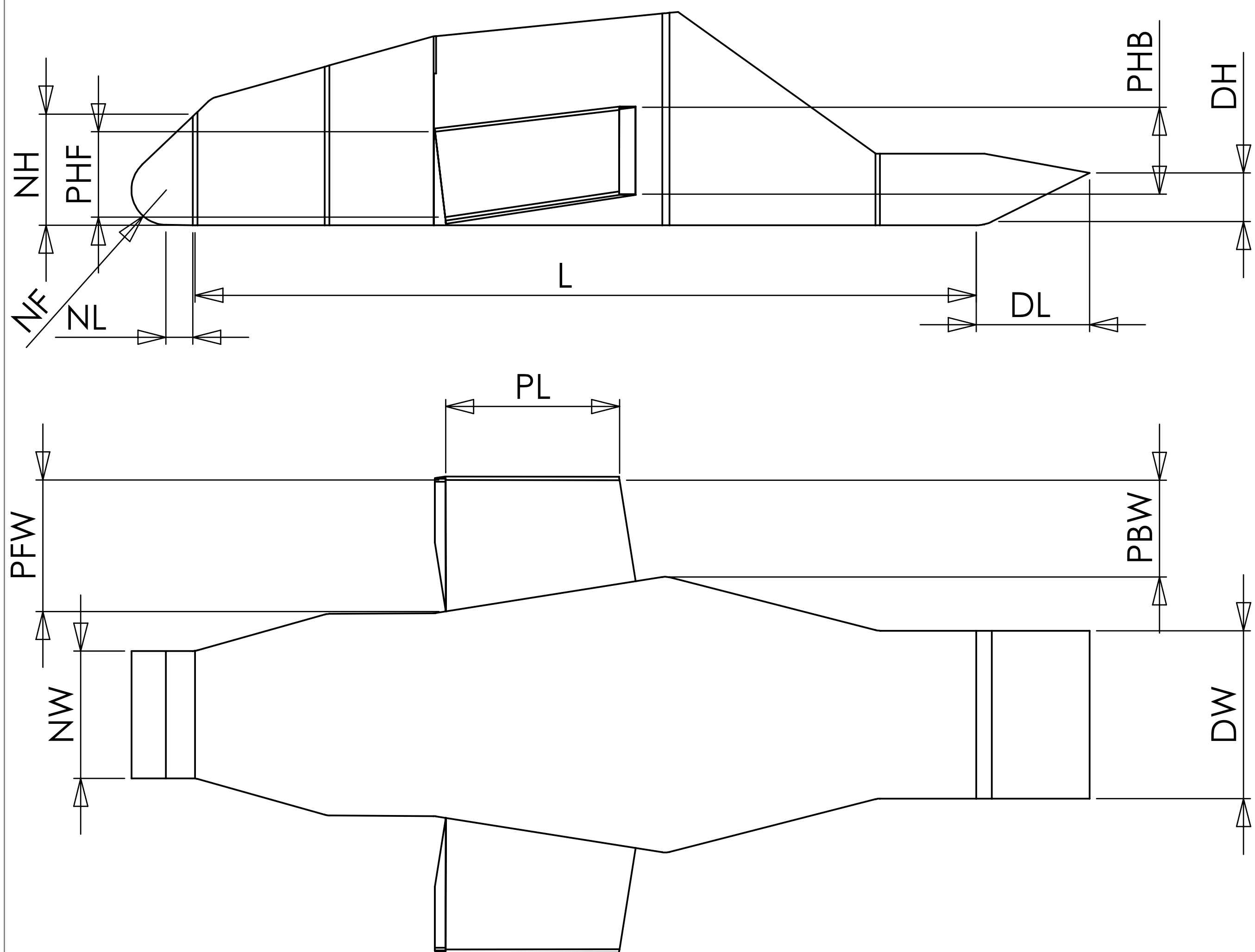
[28] Shell
http://www.shell.com
Last visit: 05/29/2013

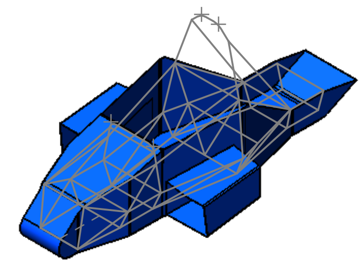
[29] 5 Facts: Vehicle Aerodynamics. Siuru, Bill.
http://www.greencar.com/articles/5-facts-vehicle-aerodynamics.php
Last visit: 05/29/2013

[30] Auto Speed. Car Aerodynamics Have Stalled. Edgar, Julian.
http://autospeed.com/cms/A_2978/article.html
Last visit: 05/29/2013

Appendix A

Blueprints



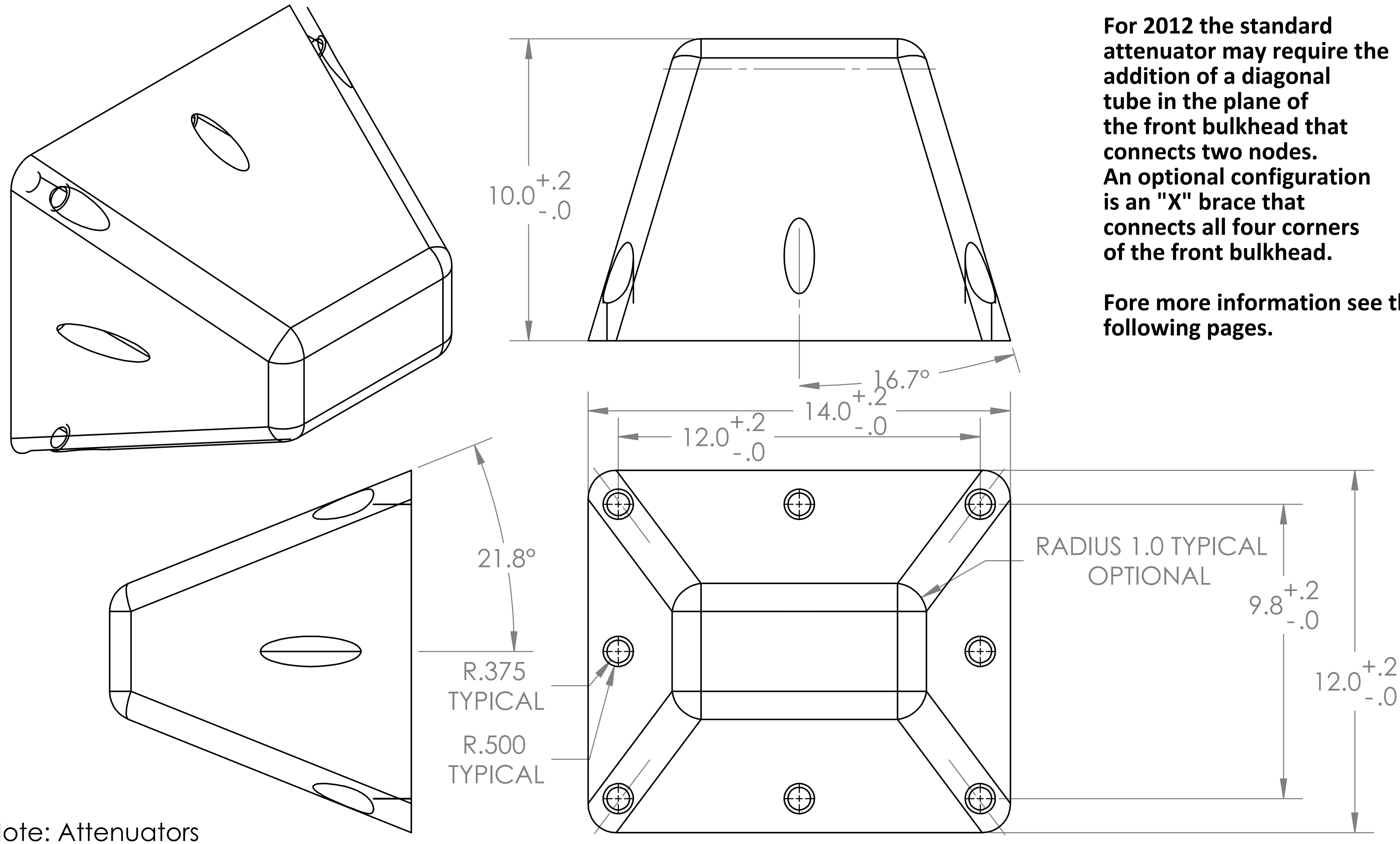


Edición de estudiante de SolidWorks.
Sólo para uso académico.

LEGEND				NAME	DATE	Universidad Carlos III
	DIMENSIONS ARE IN MM TOLERANCES: FRACTIONAL 0.015 ANGULAR: MACH 0.3 TWO PLACE DECIMAL 0.010 THREE PLACE DECIMAL 0.003		DRAWN	D. Gomez	05/29/13	Formula Student UC3M Prototype 1 Bodywork
			CHECKED			
			ENG APPR.			
			MFG APPR.			
	MATERIAL	Glass fiber	Q.A.			
NEXT ASSY	USED ON	FINISH --	COMMENTS: N = NOSE D = DIFFUSOR P = PONTOON F = FRONT B = BACK	L = LENGTH W = WIDTH H = HEIGHT		SIZE A · DWG. NO. FS-UC3M-AERO-01 · REV. 1
APPLICATION		DO NOT SCALE DRAWING				SCALE:1:20 · WEIGHT: · SHEET 1 OF 1

Appendix B

Impact attenuator



For 2012 the standard attenuator may require the addition of a diagonal tube in the plane of the front bulkhead that connects two nodes. An optional configuration is an "X" brace that connects all four corners of the front bulkhead.

Fore more information see the following pages.

Note: Attenuators purchased from BSCI will not include optional radii on edges or mounting holes. Mounting holes shown are for reference only - attachment is up to the team to determine.

Updated	11/19/11	UNLESS OTHERWISE SPECIFIED:		NAME	DATE
		DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ±0.015 ANGULAR: MACH ± 0.3 TWO PLACE DECIMAL ± 0.010 THREE PLACE DECIMAL ± 0.003	DRAWN	WBR	3/14/11
			CHECKED		
			ENG APPR.		
			MFG APPR.		
		INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.		
		MATERIAL Impax 700 Foam	The Type 12 attenuator is unchanged from the 11 but for 2012 the additional front bulkhead diagonal tube is required, see note.		
NEXT ASSY	USED ON	FINISH			
APPLICATION		DO NOT SCALE DRAWING			

TITLE: FSAE Impact Attenuator Type 11		
SIZE A	DWG. NO. **FSAE-IA-12**	REV 1
SCALE: 1:8	WEIGHT:	SHEET 1 OF 1

5 4 3 2 1